

**AutoCAD® 2008  
3D Modeling  
Workbook**

FOR

**DUMMIES®**

**by Lee Ambrosius**



Wiley Publishing, Inc.



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3D Modeling  
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Published by  
**Wiley Publishing, Inc.**  
111 River Street  
Hoboken, NJ 07030-5774

[www.wiley.com](http://www.wiley.com)

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Published by Wiley Publishing, Inc., Indianapolis, Indiana

Published simultaneously in Canada

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Library of Congress Control Number: 2007926005

ISBN: 978-0-470-09763-2

Manufactured in the United States of America

10 9 8 7 6 5 4 3 2 1





## About the Author

**Lee Ambrosius** is a former resident of a cubicle community, which he resided in for about 8 years until deciding to go into business for himself. In 2005, Lee decided to venture off into the CAD industry as an independent consultant for HyperPics, LLC in De Pere, Wisconsin, and on the Web at [www.hyperpics.com](http://www.hyperpics.com). He has been using AutoCAD since 1994 when he was first exposed to Release 12 for DOS, and has been customizing and programming AutoCAD since 1996. Lee has been an AutoCAD consultant and trainer for over a decade, and is both an Autodesk Authorized Author and an Autodesk Authorized Developer.

Over the last decade, Lee has authored a variety of works that include articles for CAD magazines, white papers for Autodesk, and has been a contributing author on a few occasions for AutoCAD books. Lee is a co-author of *AutoCAD and AutoCAD LT All-in-One Desk Reference For Dummies*. He has done technical editing for the three most recent editions of *AutoCAD For Dummies*, and the four most recent editions of *AutoCAD and AutoCAD LT Bible*. *AutoCAD 2008 3D Modeling Workbook For Dummies* is his first solo venture as a book author.

## Dedication

To my parents, for believing in me and being there when I needed them the most, especially when helping a broke college student purchase his first computer those many years ago (even though you thought it wasn't the best thing to do at the time).

## Author's Acknowledgments

I have to give a special thanks to Kristina, my wife, and my three kids, Isaac, Amber, and Chloe, for letting me put in the long hours needed to complete this book. Thanks to all the great folks at Wiley for being supportive through this entire project and giving me this great opportunity to write this book.

Thanks to Shaan Hurley, Bud Schroeder, and many of the other great professionals at Autodesk for taking the time to answer questions as they came up during this project. And following close behind, and picking up the loose ends for me, is my technical editor Mark Douglas. Mark is definitely not a stranger to AutoCAD, and is an up-and-coming individual in the CAD community who is always willing to help out a fellow user in need.

## **Publisher's Acknowledgments**

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# Introduction

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**A**utoCAD is different from many other applications that you might already know and use everyday, and that's primarily due to how it has evolved over the past two decades. The CAD industry is always striving for change in an attempt to improve the processes that are used to create drawings. (CAD stands for *Computer-Aided Drafting* or *Computer-Aided Design*, depending on whom you ask.) One of the latest trends that has started to take off and redefine the industry in recent years is 3D modeling.

Although AutoCAD is primarily thought of as a CAD application used for 2D designs, it has a powerful 3D modeling engine that allows you to create 3D models of buildings or small parts used in a machine. 3D modeling in AutoCAD was introduced over a decade ago with AutoCAD R13 (and even before that as an add-on called Autodesk Modeling Extension for AutoCAD R12), but creating 3D models was very time consuming. In recent releases of AutoCAD, the 3D modeling features have been completely overhauled to allow conceptual designing in AutoCAD and to create high-quality presentation renderings with less effort.

AutoCAD 2008 not only gives you the 2D drafting tools that you've come to use everyday, but it also provides a robust set of tools for 3D modeling and visualizing the 3D models you can create. This workbook helps you get up to speed fast with the 3D modeling features of AutoCAD, allowing you to be more productive in a shorter period of time than if you tried to go it alone.

## About This Book

*AutoCAD 2008 3D Modeling Workbook For Dummies* gives you an understanding of all the features that you need to know in order to work efficiently with 3D models in AutoCAD. This book is different from other *For Dummies* books you might have read; it provides the information that you need to know in order to work in 3D and gives you opportunities to try out what you discover throughout the book.

The book is organized to allow you to work with procedural- and workflow-based exercises. The procedural-based exercises allow you to try out the concepts covered in the book, whereas the workflow-driven exercises give you an idea of how the concepts in the book are applied when working on a 3D model.

## Conventions Used in This Book

When you need to type text at the command line, in a dynamic input tooltip, in a text box, or any other place, the text appears in **bold typeface** (for example, "Enter **perspective** at the command line"). Examples of AutoCAD prompts appear in a special typeface.

At times, you might see something like the phrase "Choose File⇨Save As." The small arrow (⇨) in this example indicates that you are to choose the File menu and then choose the Save As menu item.

## *Foolish Assumptions*

You should know how to use the Windows operating system and understand the basics of navigating folders and starting applications. To take advantage of everything that is in this book and the features that AutoCAD offers, you need at least a dial-up Internet connection (preferably a high-speed cable or DSL connection). As long as you have AutoCAD installed on the computer in front of you and a connection to the Internet, you're ready to get started.

This book also assumes that you understand how to work with the 2D drafting features of AutoCAD. If you need to brush up on your 2D skills, pick up *AutoCAD & AutoCAD LT All-in-One Desk Reference For Dummies*.

## *How This Book Is Organized*

The following sections describe the different parts that this book is organized into.

### *Part I: Introducing AutoCAD*

Part I covers working with the user interface elements in AutoCAD and some of the basic file management concepts that you should be familiar with before creating and modifying 3D models. Chapter 1 focuses on the different user interface elements that AutoCAD offers for changing settings and accessing commands related to 3D modeling along with getting help when you might need it. Chapter 2 covers some basics of working with drawing files, and Chapter 3 covers customizing the AutoCAD user interface to make sure you have access to the commands that you want to use when you need them.

### *Part II: Going from 2D to 3D*

Part II helps you transition your skills working on 2D drawings in AutoCAD to their 3D equivalent. Chapter 4 provides an overview of the different types of 3D models that you can create with AutoCAD and how to specify 3D coordinate values. Chapter 5 explains how to navigate around a 3D model as well as customizing how a 3D model is displayed in the current view so you can better see how it looks. Chapter 6 takes a look at the different drafting aids that you can use when modeling in AutoCAD, and it covers some of the basic 3D objects that you can create. You also find out how to use the general 3D modifying commands that you can use with 3D objects. Chapter 7 gives you an opportunity to apply some of the topics covered in the other chapters from this part in a workflow-driven approach to modeling in AutoCAD.

### *Part III: 3D Modeling — Solid Modeling*

Part III focuses on modeling with 3D solids. Chapter 8 explains how to create 3D solids as primitive objects such as boxes, cones, and cylinders, and it explains how to create complex 3D solids by using some of the available operations like sweeping and lofting. Chapter 9 focuses on modifying 3D solids and how to analyze your 3D models to make sure that parts don't interfere or intersect with each other. Chapter 10 provides information on techniques that allow you to create 2D views from a 3D model to create a set of shop drawings to manufacture the parts or illustrations that show how a part is assembled. Chapter 11 offers additional exercises that cover many of the topics from



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the other chapters in this part in a workflow-driven approach to working with solid models in AutoCAD.

## ***Part IV: 3D Modeling — Surfaces***

Part IV focuses on modeling with 3D faces, meshes, and surface objects to create 3D surface models. Chapter 12 covers how to create 3D faces, planar meshes, mesh primitives, and planar surface objects. Chapter 13 focuses on the creation of complex meshes and surface objects using operations like revolving, lofting, and sweeping. Chapter 14 explains how you can modify 3D faces, meshes, and surface objects after you create them. Chapter 15 offers additional exercises that cover many of the topics from the other chapters in this part in a workflow-driven approach to working with surface models in AutoCAD.

## ***Part V: Visualizing the Design***

Part V takes you through the process of adding lighting and materials to a 3D model before creating a photorealistic rendering. Chapter 16 covers creating various types of lights in your models to show contrasts on surfaces with illuminated and shaded areas. Chapter 17 focuses on creating and applying materials to 3D objects to give them realistic colors and textures that represent how the objects in the model might look after they're manufactured and assembled. Chapter 18 introduces you to creating a rendering from your 3D model after you add lights and materials to it. You also find out about some of the advanced rendering features in Chapter 18 along with using the animation features that come with AutoCAD. Chapter 19 contains additional exercises that reinforce the topics covered in the other chapters from this part.

## ***Part VI: Model Interchange***

Part VI explains many of the different ways that you can share designs with others. Chapter 20 focuses on creating layouts with floating viewports to create plotted views of your 3D models and to configure a plotter to output raster image files. Chapter 21 introduces you to the different file formats that you can exchange your 3D models in for both CAD and non-CAD users.

## ***Part VII: The Part of Tens***

Part VII provides additional information that extend your experience with AutoCAD and 3D modeling beyond this book. Chapter 22 provides an overview of the new enhancements in AutoCAD 2008 that are related to both 2D drafting and 3D modeling. Chapter 23 is packed full of links to AutoCAD Web sites that cover 2D drafting and 3D modeling, along with links to industry-related Web sites where you can find information about software add-ons to AutoCAD that can be used for analyzing 3D models or sites with 3D models that you can use in your own models.

## ***Appendixes***

Appendix A provides a listing of some of the commonly used terms that are introduced in this book, and it helps you to speak the 3D modeling lingo like a seasoned veteran. Appendix B provides you with information about the files and videos that are on the DVD that comes with this book.

## The DVD-ROM

The DVD-ROM that accompanies this workbook plays an intricate part in the workbook experience. All the files for completing the exercises are contained on the DVD-ROM along with the videos of select exercises that help explain the steps of an exercise if you get stuck.

## Icons Used in This Book



This book uses the following icons to denote paragraphs that are of special interest:

This icon indicates a question-and-answer pairing that helps tie the concepts covered under a topic with how you might use the concepts in the software.



This icon indicates information that can help you complete a task faster or help you not fall too far from the path of success.



This icon indicates a video is available for an exercise. If you need some help completing an exercise, you can watch the video to see how the exercise is completed step by step.



This icon gives you a dash of extra information if you haven't had enough caffeine yet for the day. AutoCAD is a complex program, and it takes time to put all the puzzle pieces together, so use these friendly reminders to stay on the right track.



This icon helps you avoid potential pitfalls in AutoCAD that can get you into trouble. Failure to adhere to the message may result in an undesired side effect to your 3D model or additional work that could have been avoided.

## Where to Go from Here

This book is organized to allow you the freedom of moving from part to part or chapter to chapter in any order that you want. However, each of the chapters build on concepts that are covered in previous chapters of the book; so if you're relatively new to 3D in AutoCAD, I recommended that you start at the front of the book and work your way towards the back. If you're experienced with AutoCAD, you might want to skim through the first three chapters of the book and then dive into Chapter 4.

This book was written with the assumption that you already know how to use the 2D drafting features of AutoCAD. Although I do cover many topics that cross over between 2D and 3D, I primarily focus on using features for 3D modeling. If you want to brush up on the latest 2D drafting features in AutoCAD or some of the drawing precision tools that AutoCAD offers, I suggest getting *AutoCAD & AutoCAD LT All-in-One Desk Reference For Dummies*.

3D modeling isn't something that you'll be an expert at overnight, but completing the exercises in this book and applying the techniques to your own projects can help you to reach that next level in CAD design. No matter where your new found 3D modeling skills take you, be sure to have fun.

# Part I

# Introducing AutoCAD

The 5<sup>th</sup> Wave

By Rich Tennant



"You know kids - you can't buy them just any multi-dimensional drafting software."

## *In this part . . .*

**A**utoCAD is a very powerful 2D drafting application, but behind the user interface (toolbars, pull-down menus, and drawing window) lie the capabilities of navigating and creating 3D models and generating presentation-quality images from 3D models to share with your clients. Before you begin navigating or creating your own 3D models from 3D solids or surfaces or using materials and user-defined lights to generate a photorealistic rendering of a 3D model, you must become familiar with how AutoCAD's user interface works in 3D. To be proficient in 3D, you must understand all the different user interface elements that are available in AutoCAD. When you understand the elements of the user interface, you can explore some of the basics of working with drawing files and how to make sure the 3D tools and commands that you want to use are accessible by customizing the layout of AutoCAD's user interface.

## Chapter 1

# AutoCAD and the User Interface

---

### *In This Chapter*

- ▶ Launching AutoCAD and using command line switches
  - ▶ Looking at the user interface of AutoCAD
  - ▶ Using commands and providing input
  - ▶ Getting help from AutoCAD
- 

**B**eing proficient with AutoCAD and its user interface can help make drafting tasks easier based on how familiar you are with all the tools that are just a click away. The exercises in this chapter give you an understanding of how to use the various interface elements that AutoCAD has to offer to make sure you have the tools available when you need them.

The exercises in this chapter use the AutoCAD Classic workspace. To set the AutoCAD Classic workspace current, choose Tools⇨Workspaces⇨AutoCAD Classic. For more information on using and creating workspaces, see Chapter 3.

## *Launching AutoCAD*

AutoCAD is one of the most popular computer-aided drafting applications on the market today. You can use it to design plans for residential homes and complex drawings such as paper-converting machines and circuit boards. To become efficient with AutoCAD, you should get to know as many methods of launching the program as possible.



1. How can you start AutoCAD?
  - A. From the Start menu
  - B. From a shortcut on the desktop, taskbar, or Quick Launch toolbar
  - C. Insert the product CD/DVD into the CD/DVD-ROM
  - D. By opening a drawing file

---

### **Exercise 1-1: Launching AutoCAD**

In this exercise, you add a shortcut to the desktop, the Start menu, and the Quick Launch toolbar.



- ✓ **Add an AutoCAD shortcut to the Windows desktop by creating a copy of the one that appears in the AutoCAD 2008 folder.** Choose Start⇨(All) Programs⇨Autodesk. Right-click the AutoCAD 2008 icon and choose Send To⇨Desktop (Create Shortcut).

- ✓ **Add a shortcut for AutoCAD to the Start menu.** Right-click the AutoCAD shortcut on the desktop and choose Pin to Start Menu.
- ✓ **Add a shortcut for AutoCAD to the Quick Launch toolbar.** Drag and drop the AutoCAD shortcut from the desktop into place on the Quick Launch toolbar. If the Quick Launch toolbar isn't visible on the taskbar, right-click the taskbar and choose Toolbars⇨Quick Launch.

AutoCAD allows you to use what are called *command line switches*. Command line switches automate some of the tasks that are normally performed each time you start AutoCAD. Removing manual steps from the start-up process ensures that the drawings you create are more consistent with each; then you can worry more about the models that you need to create than all the intricate setup processes. You use them in conjunction with desktop shortcuts that define the default template, workspace, and user profile that are set up when AutoCAD first starts.

<b>Command Line Switch</b>	<b>Description</b>
/p	Allows you to specify which user profile to use when AutoCAD starts. For information on user profiles, see Chapter 3.
/t	Allows you to specify which drawing template to use for the default drawing that is created when AutoCAD launches. For more information on drafting templates and creating a new drawing, see Chapter 2.
/w	Allows you to specify which workspace to use when AutoCAD starts. For information on user workspaces, see Chapter 3.

You can use more than one command line switch at a time with a shortcut; you just need to use them correctly. Figure 1-1 shows an example of using the /t and /w command line switches.



**Figure 1-1:**  
The  
AutoCAD  
2008  
Properties  
dialog box.



2. In Exercise 1-2, you create a command line switch for AutoCAD. To access the shortcut properties in order to add a command line switch, right-click and choose \_\_\_\_\_.

### Exercise 1-2: Customizing a Desktop Shortcut with Command Line Switches

In this exercise, you customize a desktop shortcut (see Exercise 1-1) to specify a template for the default drawing and which workspace should be set current when AutoCAD is first launched.

1. Access the Properties dialog box for your AutoCAD shortcut.
2. In the Properties dialog box, click the Shortcut tab.
3. On the Shortcut tab, position the cursor at the end of the path to the `acad.exe` file in the Target text box and click.
4. Press the spacebar once to add a space after the executable file's path. Then enter this text:

```
/t acad3d.dwt /w "3D Modeling"
```

The `/t` command line switch uses the drawing template `acad3d.dwt` that ships with AutoCAD to create the default drawing, and the `/w` command line switch sets the 3D modeling workspace current. After you add the text in the Target text box, the text looks like the following:

```
"C:\Program Files\AutoCAD 2008\acad.exe" /t acad3d.dwt /w "3D Modeling"
```

5. Click the General tab and change the name of the shortcut in the Name text box near the top to **My AutoCAD**.
6. Click OK to save the changes.
7. Double-click the shortcut to test the changes made.

AutoCAD launches with the default drawing based on the `acad3d.dwt` file and the 3D Modeling workspace is set current.

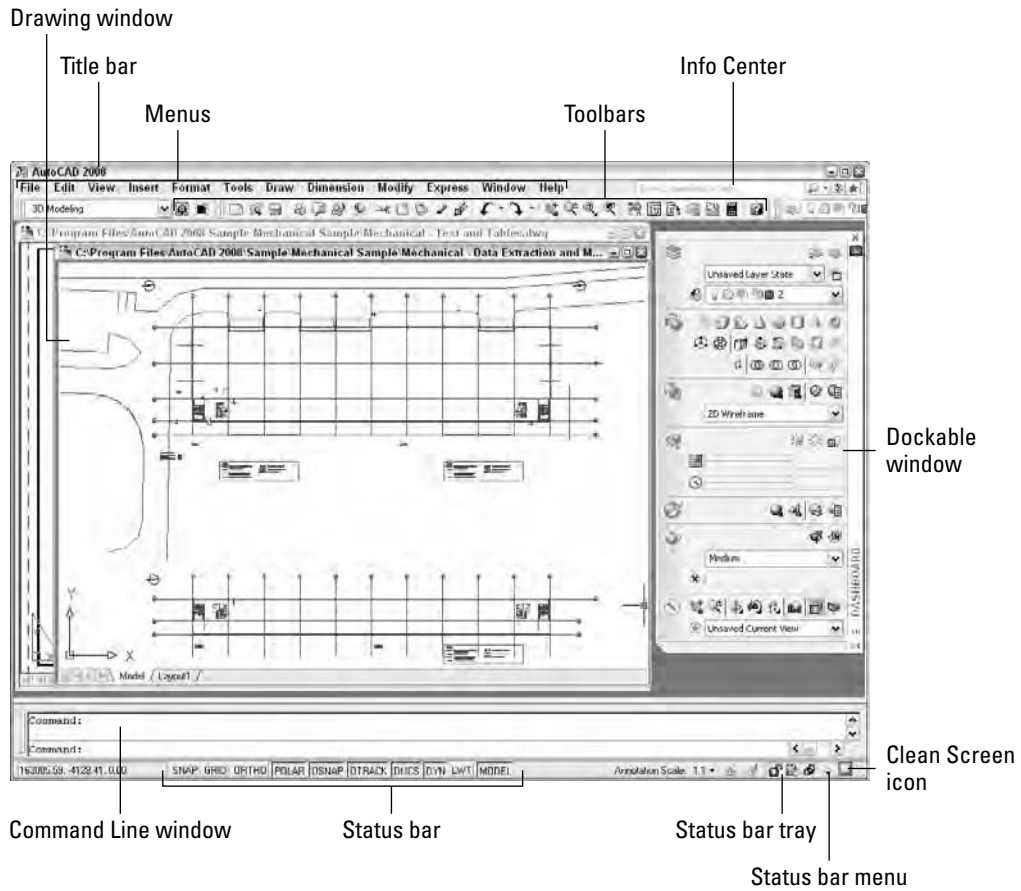


If the name of the drawing template, profile, or workspace contains a space, you must wrap the name with double quotation marks in order for AutoCAD to interpret the command line switches properly.

For additional information on other command line switches that are available for use with an AutoCAD shortcut, see the Customize Startup topic in the AutoCAD Online Help system.

## Navigating the User Interface

The AutoCAD user interface — or *UI* as it's often referred to — can be very overwhelming, but it doesn't have to be. All you need is some understanding of how things are laid out and what the various UI elements are. AutoCAD, like other Windows-based applications, has pull-down menus, toolbars, a document area, and a status bar area, but AutoCAD also has some unique user interface elements that aren't found in a lot of other applications. Some of the unique interface elements are a Command Line window, dynamic input tooltips, and dockable windows. Figure 1-2 shows many of the different user interface elements that are found in AutoCAD.



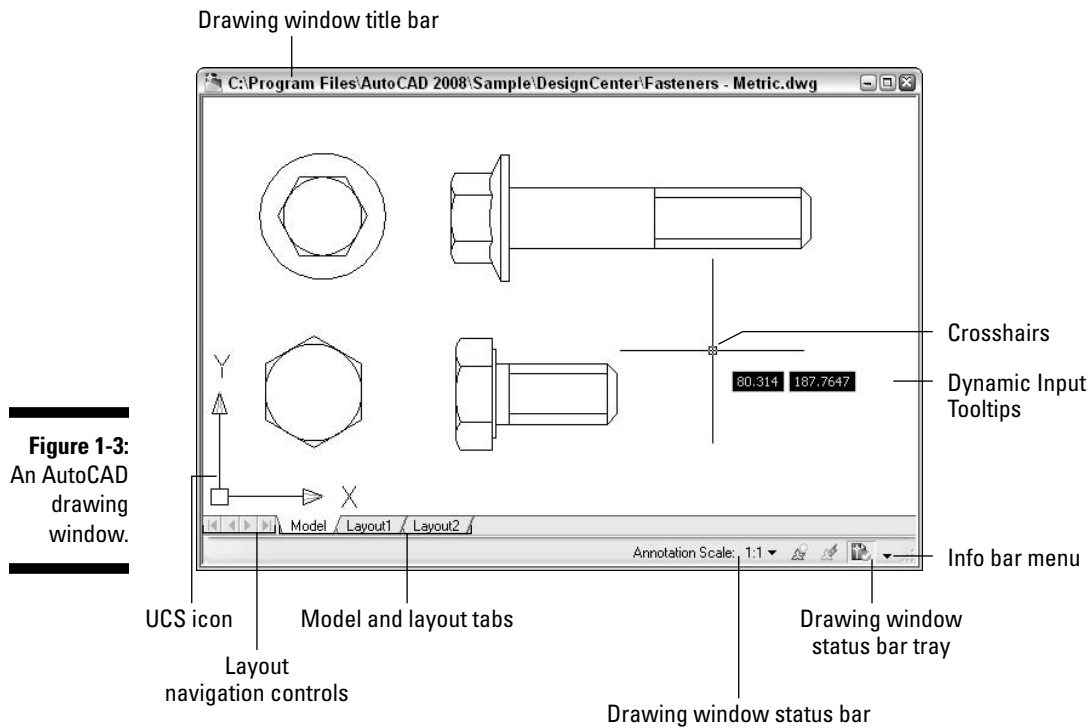
**Figure 1-2:**  
An AutoCAD  
application  
window.

## Drawing window

The *drawing window* (see Figure 1-3) is the main area in the middle of AutoCAD where you add and modify objects in the designs that you create. When you create a new drawing or open an existing drawing, a drawing window appears. Each open drawing has its own drawing window. Along the bottom of each drawing window, you find these items:

- ✓ **A series of tabs** similar to those found in a spreadsheet program; these tabs represent where you create your model (Model tab) and the layouts used for plotting different views of your model.  
You can turn off the Model tab and layout tabs if you don't want them displayed. In their place are two buttons that allow you to switch to a different layout tab.
- ✓ To the left of the tabs are **navigation controls** that allow you to scroll to the first, last, next, or previous layout tab if all the tabs can't be displayed along the bottom of the drawing window at a time.
- ✓ **The drawing status bar** along the bottom displays settings specific to the drawing, such as the current annotation scale. If you disable the drawing status bar, the options are displayed in the status bar area of the application window instead.





### Exercise 1-3: Navigating the Drawing Window

Practice using some of the user interface elements of the drawing window.

1. Create a new drawing based on the ANSI A (portrait) -Color Dependent Plot Styles.dwt template that comes with AutoCAD.



In AutoCAD, choose File⇒New. In the Select Template dialog box, select the ANSI A (portrait) -Color Dependent Plot Styles.dwt template file and click Open. The drawing templates that come with AutoCAD are located in the Templates folder at C:\Documents and Settings\user name\Local Settings\Application Data\Autodesk\AutoCAD 2008\R17.1\enu; substitute *user name* with your Windows login name.

Turn to Chapter 2 to find out how to create a new drawing.

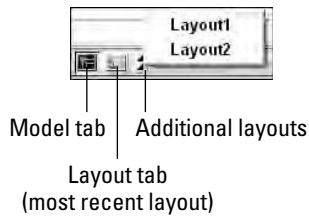
2. Click the Model tab and layout tabs to view the contents of each one.
3. Right-click one of the tabs and choose Hide Layout and Model Tabs to toggle the display for the layout tabs.

To display the tabs again, right-click the Model or Layout button in the status bar area and choose Display Layout and Model Tabs.

4. View the contents of each of the layouts by clicking the Model and Layout buttons in the application window's status bar; they're displayed to the right of the button labeled LWT (as shown in Figure 1-4).
5. Click the Status Bar Menu button to the left of the Clean Screen icon on the application status bar (refer to Figure 1-2). Choose Drawing Status Bar. Make sure that a check mark is next to Drawing Status Bar.

Some of the icons in the status bar tray move to the bottom of the drawing window along with the controls for annotation scale.

**Figure 1-4:**  
AutoCAD  
status bar  
with Model  
and Layout  
buttons  
displayed.



## 6. Turn off the Drawing Status Bar setting.

Click the button to the right of the Layout button named Additional Layouts to display a list of all the layouts in the drawing.

Press Ctrl+Page Up to navigate to the previous layout or Ctrl+Page Down to navigate to the next layout in the drawing.



## Command Line window

The Command Line window by default is displayed between the drawing window and the status bar area. If you don't like its position, you can dock it above or along the left or right side of the drawing window. The Command Line window plays a key role in inputting commands, options, and values to create and modify objects; it also displays command prompts and messages to let you know what AutoCAD is waiting for.



If you want to gain additional room in the AutoCAD application window, you can close the Command Line window. Press Ctrl+9 to toggle the display of the Command Line window on and off. When the Command Line window is hidden, command prompts and options are displayed in dynamic tooltips. Press F12 to toggle the display of dynamic tooltips. For more information on dynamic input, see the "Dynamic input" section later in this chapter.

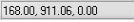













## Text window

The Text window displays a running history of the commands and options that you previously entered in the Command Line window or at the dynamic input tooltip. You won't need the Text window very often, but if you need to see what you did a few commands ago, press F2 to bring up the Text window.

## Status bar

The status bar allows you to quickly access drafting settings and other settings that are used for some of the features. Most of the settings on the status bar allow you to toggle drafting settings on or off without opening a dialog box or remembering a shortcut key combination. Also, you can right-click most buttons to access options that allow you to change the behavior of the drafting settings.

Table 1-1 lists the different buttons on the status bar.

<b>Table 1-1 Status Bar Buttons</b>		
<b>Button</b>	<b>Name</b>	<b>Description</b>
	Coordinates	Toggles among absolute, relative, and polar coordinate display settings
	Snap Mode	Toggles grid or PolarSnap on and off
	Grid Display	Toggles the display of the grid on and off
	Ortho Mode	Toggles ortho mode on and off
	Polar Tracking	Toggles polar tracking on and off
	Object Snap	Toggles running object snaps on and off
	Object Snap Tracking	Toggles object snap tracking on and off
	Dynamic UCS	Toggles dynamic UCS on and off
	Dynamic Input	Toggles dynamic input on and off
	Show/Hide Lineweight	Toggles the display of lineweights on and off
	Model/Paper Space	Toggles between model and paper space
	Model/Layout and Additional Layouts	Toggles between model and paper space, and switch paper space layouts
	Minimize/Maximize and Viewport Navigation	Toggles between maximized and minimized viewport states, and allows you to switch among different viewports
	Status Bar Tray	Displays the current notification icons for the features that are in use in the current session and drawing

#### Exercise 1-4: Using the Status Bar

In this exercise, you create a new drawing and use some of the drafting settings that are available on the status bar.

1. Choose File→New.
2. In the Select Template dialog box, double-click the `acad.dwt` drawing template file.

3. Choose Draw↔Line, or type **line** at the command prompt and press Enter.
4. Pick a point in the drawing window to start drawing a line.
5. On the status bar, click DYN until the button appears popped out. When a button is popped out, the drafting setting is turned off, and in this case you toggled Dynamic Input off.

Move the crosshairs in the drawing window; no dynamic input tooltips are displayed trailing the crosshairs. The dynamic input tooltip is an extension of the Command Line window that displays the current command prompt and input boxes that allow you to specify coordinate values, distances, or angles among other values for the current command. For more information on dynamic input, see the “Dynamic input” section later in this chapter.

6. Turn on Dynamic Input.

Move the crosshairs in the drawing window; the dynamic input tooltips display trailing the crosshairs. With the LINE command active yet, you should see the command prompt *Specify next point* and a down arrow that allows you to access the available command options. After the command prompt, you see two input boxes that allow you to specify the X- and Y-coordinate values for the end point for the line.

7. Turn on ortho mode.

Move the crosshairs in the drawing window; you can draw a straight line only in 90-degree increments starting with 0 because ortho mode constrains the coordinate value horizontal or vertical from the previous point specified.

8. Turn off ortho mode.

You can now create a line at any angle that you want to.

9. Press Esc to exit the LINE command.

---

### Exercise 1-5: Controlling the Display of Controls on the Status Bar

For this exercise, open a drawing (a new one or one from a previous exercise). Practice controlling the display of controls on the status bar.










1. On the right side of the status bar area, click the Status Bar Menu button (located to the left of the Clean Screen icon).
2. Select the controls that you want to display or hide on the status bar.

The items selected are displayed, and the ones that aren't selected are currently hidden. It's a good idea to display only the controls that you frequently use.

## Status bar tray

On the right side of the status bar is an area called the *status bar tray*. The status bar tray provides access, via an icon, to features that you're currently using in the active drawing or AutoCAD. When the icon for a feature is displayed, you can click or right-click the icon to access settings and other options for the feature. As you work in a drawing, certain situations cause a notification balloon to be displayed. The notification balloon lets you know when you've done something that affects your drawing, such as adding a new layer that doesn't match the current set of CAD standards or completing a background plot. You can control the display of many of the icons or toggle on and off the display of the status bar tray.

Table 1-2 lists the different tray icons that are available in AutoCAD.

<b>Table 1-2</b>		<b>Status Bar Icons</b>
<i>Button</i>	<i>Name</i>	<i>Description</i>
	Toolbar/Window Lock	Controls whether toolbars and dockable windows (palettes) can be moved. Locking/unlocking can affect all toolbars or dockable windows. You can specify which are floating or docked.
	CAD Standards Notification	Identifies that a drawing has CAD Standards associated with it. Access the tools related to CAD standards and receive notifications when the drawing contains standards violations.
	External Reference Notification	Identifies that External Reference drawings are attached. Receive notifications when the files have been updated and need to be reloaded.
	Plot Notification	Accesses information about recently plotted drawing files.
	Attribute Extraction Notification	Identifies that a table object in the drawing is linked to the attributes of blocks in the drawing and receive notifications when the table object becomes out of date.
	Trusted Autodesk DWG	Indicates whether the drawing was created with an application developed by Autodesk or using RealDWG.
	Clean Screen	Toggles the state of Clean Screen.
	Data Link	Indicates if data in a table is linked to an external data source.
	Vault	Indicates whether you are currently logged into Autodesk Vault. The icon is available only when Autodesk Vault is installed on your computer.

### Exercise 1-6: Controlling the Display of Icons and Notification Balloons

For this exercise, open a drawing (a new one or one open from a previous exercise). Practice changing the global settings for the status bar tray and notification balloons.

1. On the right side of the status bar, click the Status Bar Menu button (located to the left of the Clean Screen icon).
2. Choose Tray Settings.
3. In the Tray Settings dialog box, click Display Icons from Services so it's deselected.
4. Click OK.

The status bar tray is now hidden, with the exception of the Status Bar Menu and Clean Screen buttons.

5. Open the Tray Settings dialog box, re-enable Display Icons from Services, and select Display Time and 5 seconds from the drop-down list.

The notification balloons now close automatically after 5 seconds.

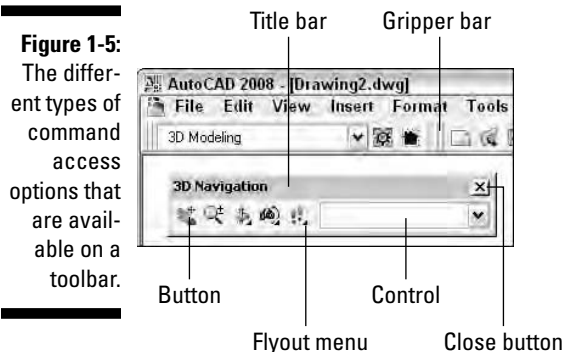
6. Click OK.

## Toolbars

Toolbars are some of the most commonly used user interface elements in a Windows-based application. Toolbars allow you to quickly access commands from an organized grouping of tools instead of using pull-down menus. A toolbar contains buttons that you click to activate a command, and it has controls that allow you to select an option from a drop-down list or enter text into a text box.

Some toolbars contain *flyouts*, which have multiple tools on them that perform similar tasks. A flyout on a toolbar is indicated by a button displaying a black arrow in the lower-right corner of the button's image.

Figure 1-5 shows the main parts of a toolbar.



You can dock toolbars along the edge of the application window, or you can have them floating. The state that the toolbar is in doesn't affect how the buttons and controls work, but if a toolbar that contains a drop-down list is docked on the left or right side of the application window, the control isn't displayed. When a toolbar is docked, the gripper bar is displayed at the top or left side of the toolbar based on how it's docked; if the toolbar is floating, a title bar appears at the top with the name of the toolbar.

Most people prefer toolbars to pull-down menus because they offer quick access to commonly used commands, and toolbar icons make the commands easy to recognize. AutoCAD comes with 37 toolbars, but you can create your own custom commands and toolbars to access the commands that you use on a regular basis with ease. For more information on creating your own custom commands and toolbars, see Chapter 3.

### Exercise 1-7: Displaying, Hiding, and Repositioning a Toolbar

For this exercise, open a drawing (a new one or one from a previous exercise). Practice displaying, hiding, and repositioning a toolbar.

1. Right-click any displayed toolbar and choose 3D Navigation from the shortcut menu to display the 3D Navigation toolbar.

Toolbars that are currently displayed have a check mark next to them on the shortcut menu, and the ones that are hidden don't.



2. If the 3D Navigation toolbar is currently docked, click the gripper bar and drag the toolbar over the drawing window.

You can lock the position of toolbars and dockable windows by clicking the Toolbars/Window Lock icon in the status bar tray.

3. Click the title bar of the 3D Navigation toolbar and drag the toolbar along the left side of the application window of AutoCAD. Release the mouse button to dock the toolbar.

When the toolbar gets close enough to be docked, the drag preview of the toolbar switches from a horizontal to a vertical orientation.



Hold down Ctrl when dragging a toolbar or dockable window to keep it from docking along the edge of the application window.

4. Close the toolbar.

## Menus

Menus — or pull-down menus — are located on the menu bar and allow you to access more commands than any other user interface in AutoCAD (other than the command prompt). You can find most of the commands that you need when working on a drawing on one of the 12 menus in AutoCAD. The menus in AutoCAD contain commands that have been organized into related groups; for instance, commands that create objects are on the Draw menu, and ones used for dimensioning are on the Dimension menu. The main menus that are available in AutoCAD are located along the top of the application window. When clicked, they appear to drop down from the main menu's name.

AutoCAD also offers context-sensitive menus that are available when you right-click the drawing window. Context-sensitive menus offer different options based on whether a command is active or even whether any objects are selected in the drawing before you right-click. Both the main and shortcut menus use submenus to organize options that are similar. Submenus are indicated by a small black arrow that points to the right.

Menu items use special indicators that let you know what happens when you click or access the item without using the menu in the future:

- ✓ **A series of 3 periods (. . .):** A dialog box displays when you choose the menu item.
- ✓ **Shortcut key combination:** The keyboard command is displayed to the right of the menu item's caption. For example, Ctrl+N is displayed next to the New menu item. You can press Ctrl+N to start the NEW command.

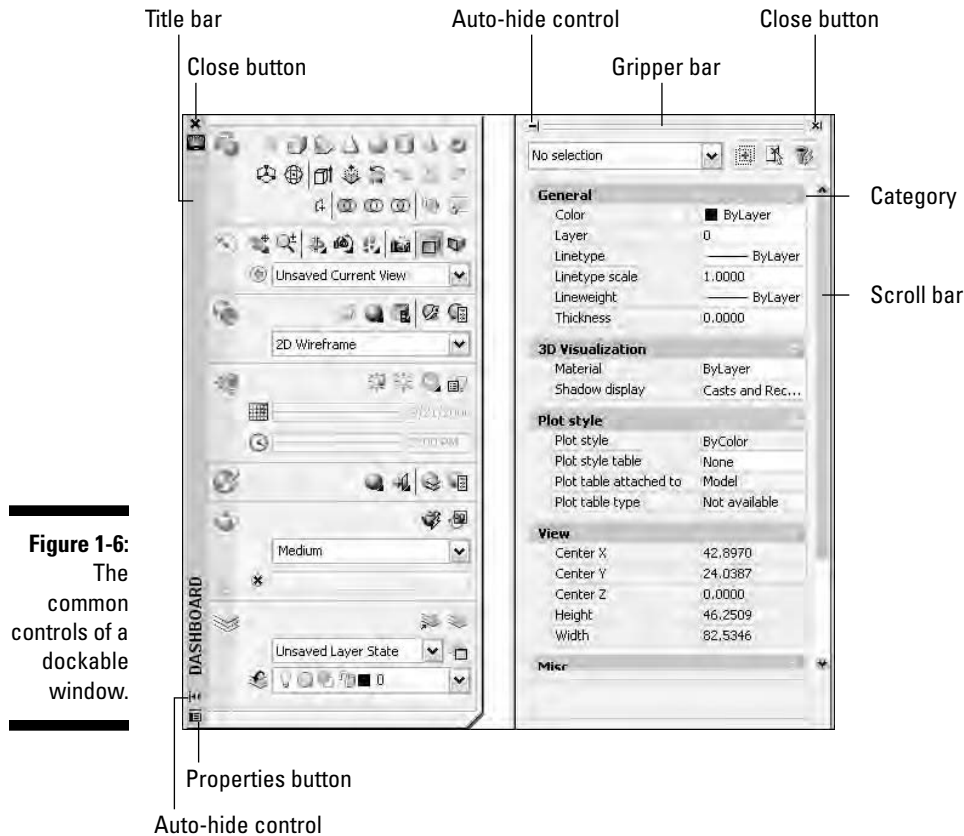


If you hold down Shift before right-clicking, you can access the Object Snap Cursor menu, which allows you to select an object snap override.

## Dockable windows

Dockable windows — or palettes — are one of the newest user interfaces in AutoCAD. They allow you to access settings and options without the need of closing them when you're done. AutoCAD offers a number of dockable windows that allow you to modify the properties of an object; calculate distances; convert values to different units of measurement; access reusable objects (blocks, materials, and visual styles) when creating or modifying a drawing; or work with sheet sets. AutoCAD comes with 14 dockable windows, and you can launch most of them by choosing Tools⇨Palettes.

You can dock, as the name implies, dockable windows or have them float like a toolbar. However, when docked or floating, a dockable window can still take up a moderate amount of space in AutoCAD, which can reduce the available space you have for the drawing window. That's why you can auto-hide dockable windows. The Auto-hide setting collapses the dockable window when the cursor isn't over it, which helps to reduce it to the same size as a docked toolbar. Figure 1-6 shows the common controls that all dockable windows have in common. Manipulating the position and display of a dockable window is similar to manipulating a toolbar.



**Figure 1-6:**  
The  
common  
controls of a  
dockable  
window.

### Exercise 1-8: Manipulating the Display of a Dockable Window

For this exercise, open a drawing (a new one or one from a previous exercise). Practice displaying, auto-hiding, and repositioning a dockable window.

1. Choose Tools⇄Palettes⇄Properties.
2. If the Properties window is currently docked, click the gripper bar and drag it over the drawing window.
3. Click the Auto-hide control on the title bar.

The dockable window collapses, and only the title bar is displayed. Moving the crosshairs/cursor over the title bar causes it to expand.

4. Click the title bar and drag the palette to the right of the drawing window. Release the mouse button when the preview of the palette changes to dock the palette.

Hold down Ctrl when dragging a toolbar or palette to keep it from docking along the edge of the application window.





5. Click the Auto-hide control (which looks like the Minimize icon) to the left of the gripper bar.

The dockable window collapses, and only the title bar is displayed. Moving the crosshairs/cursor over the title bar causes it to expand.

6. Close the dockable window.

Click the X button on the dockable window or choose Tools⇨Palettes⇨Properties.

## Giving AutoCAD Directions

As you might have noticed by all the different user interfaces that AutoCAD offers, you can start commands and input options in a number of ways. You can use commands to start a specific task that displays prompts or a dialog box for input.

The easiest way to start a command is to type it in the Command Line window. After you start a command, you do one of the following:

- ✓ Provide input if a prompt is displayed in the Command Line window or at a dynamic input tooltip.
- ✓ Provide input in the controls of a dialog box that is displayed.
- ✓ Nothing; the command simply ends because no input is required.



3. Besides typing the command at the command prompt, how else can you start a command?
  - A. Choose a menu item from a pull-down menu or shortcut menu.
  - B. Use the Command Prompt window in Windows.
  - C. Use a toolbar or the dashboard.

## It's all in the name

Not all commands in AutoCAD are created equally: You can use some commands when other commands are in use, whereas you can use certain commands only when no other command is active. When you're using a command that creates or modifies an object, the command is usually a modal command that supports being used when another command is active. To use a command, you simply enter its name at the command prompt and press Enter or select the associated user interface element to start a command.



When entering command names and options at the command prompt, press the spacebar in most cases to start a command or option. The spacebar acts like the Enter key unless the input that is being requested contains a space.

### Transparent commands

You can use some commands in AutoCAD when a command is already active; these commands are known as *transparent commands*. (You can also use transparent commands when no command is active.) To use a command transparently, enter an apostrophe (') in front of the command's name. Transparent commands affect the properties of a new object when it's being created or affect the display of the drawing.

For the most part, you don't need to worry about entering an apostrophe (') in front of the command's name when using a menu or toolbar to start a command because it's part of the macro for the user interface element.

### ***Command alias***

AutoCAD uses *command aliases* as an option to starting commands. A command alias is an abbreviation that is assigned to a command. It's used to access a command from the command prompt without having to enter its full name. Command aliases are typically one to three characters in length, but some aliases are longer. Examples of some aliases are L for the LINE command and 3DO for 3DORBIT. Command aliases are stored in a file named `acad.pgp` and are loaded automatically when you start AutoCAD.

Although AutoCAD ships with a number of command aliases, you can add your own command aliases. They're stored in the `acad.pgp` file. To open it, choose Tools ⇨ Customize ⇨ Edit Program Parameters (`acad.pgp`). For additional information on creating and modifying command aliases, see the Define Custom Commands topic in the Online Help system of AutoCAD. After modifying the `acad.pgp` file, you have to either close and restart AutoCAD for the changes to take effect or use the REINIT command to reload all the aliases in the `acad.pgp` file without restarting AutoCAD.



If you don't remember the full name of a command but know the first few letters of it, you can enter those at the command prompt and then press Tab until you find the command you are looking for. After the command's name is displayed at the command prompt, press Enter to use it.

## ***Repeating a command***

When you're working in a drawing, you sometimes want to use the same command multiple times in a row. AutoCAD allows you to repeat the previous command without having to select it from a user interface element again, making it much more efficient to continue using the command.

You can use these methods to repeat a previously used command:

- ✓ Press Enter when no command is active.
- ✓ Press the spacebar when no command is active.
- ✓ Press the Up arrow key when no command is active and press Enter. You can step back to retrieve previously used commands within the current session and not just the most recently used command.
- ✓ Right-click the drawing window and choose Repeat *Command Name*.
- ✓ Right-click the drawing window and choose Recent Input ⇨ *command\_name*.
- ✓ Right-click the Command Line window and choose Recent Commands ⇨ *command\_name*.

## ***Canceling the current command***

Sometimes you might find yourself using a command, and all of a sudden you want to stop using it and switch to a different one. If you don't want to complete the current command, press Esc or right-click and choose Cancel from the contextual menu.



Before you press Esc, make sure that you don't want to complete the current command. When you press Esc, you might lose some of the changes that you made.

---

### Exercise 1-9: Starting and Canceling a Command

For this exercise, create a new drawing and set the AutoCAD Classic workspace current. Then practice starting and canceling commands from a menu, toolbar, and command prompt.

1. Choose Draw→Modeling→Box.
2. Click a point in the drawing window.
3. At the command prompt, enter **pan** and press Enter.
4. Click in the drawing window and drag in any direction to pan. When you're finished panning, press Enter to resume the BOX command.
5. Right-click and choose Cancel to end the BOX command.

The BOX command exits without adding a new object to the drawing.

6. Click the Erase icon on the Modify toolbar.
7. Press Esc to cancel the ERASE command.
8. At the command prompt, enter **co** and press Enter.

CO is the command alias for the COPY command, so the COPY command starts.

9. Exit the COPY command with the method of your choice.
10. Repeat the last command with the method of your choice.

Press Enter or the spacebar or use one of the options from a shortcut menu.



## Dynamic input

*Dynamic input* allows you to get prompts near the location of the crosshairs without having to look up and down between the Command Line window and the drawing window. Not only does dynamic input allow you to save on some neck strain, but it keeps your focus in the drawing window where you do most of your work inside AutoCAD.

Based on the current settings of dynamic input when it is enabled, AutoCAD provides feedback to you on the current location of the crosshairs when no command is active and then provides multiple ways to specify coordinates and values when certain commands are active. For example, when the LINE command is active, dynamic input tooltips allow you to input the X and Y coordinate for the line object to start at. Then when the first point is selected, two dynamic input dimensions are displayed: one for the distance of the line and the second for the angle of the time. For more information on changing the way dynamic input works, see Chapter 3.

## Dialog boxes

Command line input and options are the most common forms of providing information to AutoCAD, but in some cases, it can be a complex process. When you can benefit from a dialog box instead of a command line, a dialog box is provided. As in many

other Windows-based applications, dialog boxes in AutoCAD collect information from you through standard controls such as text boxes, radio buttons, check boxes, and drop-down lists. You can display dialog boxes in two different states: modal and modeless. *Modal* dialog boxes don't allow you to do anything else in AutoCAD until the dialog box has been closed; a *modeless* dialog box allows you to interact with AutoCAD while it's displayed. Dockable windows (palettes) are a form of a modeless dialog box.

Many commands display a dialog box as the primary form of input, but they also allow you to input information using the command prompt. In most cases, adding a hyphen (-) in front of the command's name switches the command from a dialog box to the command prompts. For example, the HATCH command displays a dialog box, but the -HATCH command displays a series of command prompts instead.

Some dialog boxes, such as the Options dialog box, don't offer a single command to access the options found in the dialog box, but rather many different commands and system variables. As an example, if you want to disable background plotting, you use the system variable BACKGROUNDPLOT because there is no -OPTIONS command. For more information on system variables and the Options dialog box, see Chapter 3.



Some commands such as NEW and OPEN don't have equivalent commands that start with a hyphen but instead rely on the system variable FILEDIA. Set FILEDIA to 0 (zero), and the commands display prompts instead of the Standard File Navigation dialog box.

## *Sending an S.O.S. to AutoCAD*

AutoCAD is a very complex application that offers a lot of different tools to help with many different general drafting tasks and some that are much more specific to an industry. With all this complexity, there are times when you need a little help to keep you on track with learning something new or trying to complete a task. The Online Help system that AutoCAD offers does a pretty good job of giving answers to both general and specific problems, but because AutoCAD is used by a very large, diverse group of people, you might not find all the answers in the Online Help system. Fortunately, because AutoCAD has a large user base, usually someone has encountered the specific problem that you have and might even have posted the solution on the Internet in one of the many online forums out there.

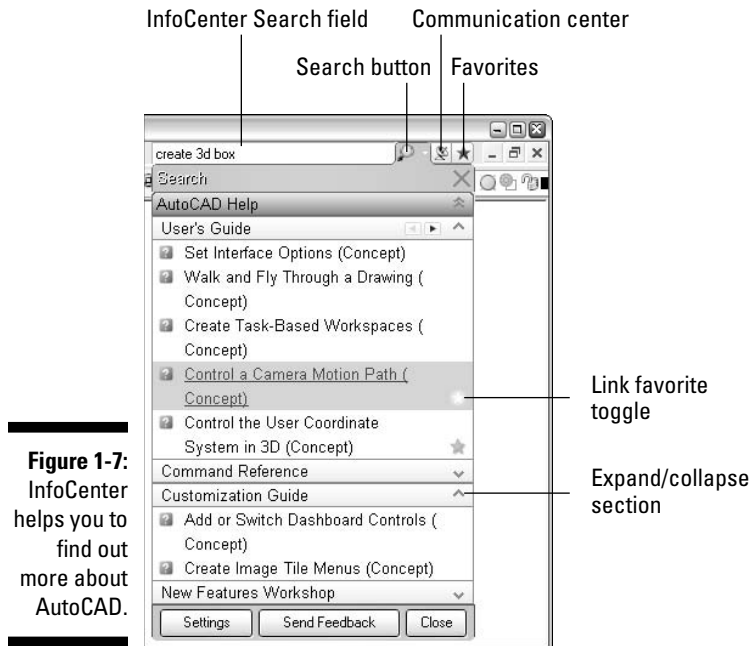
### *Online help*

The Online Help system that comes with AutoCAD is very complete for the most part, with a few minor potholes here and there. The Online Help system is well integrated in AutoCAD, and you can access it just about anywhere in the application to get context-sensitive help when you need it most. To access help, choose Help⇨Help or enter **help** at the command prompt.

### *InfoCenter*

InfoCenter (see Figure 1-7) replaces the Info Palette from previous versions and combines the functionality of the Communication Center in one place. The InfoCenter is located in the upper-right corner of AutoCAD on the menu bar and allows you to search for information in the Online Help system as well as get information from your favorite blogs or Web sites that post content in an RSS (Really Simple Syndication) file

format. If you find a specific topic or entry of interest, you can flag it as a favorite, which allows you to easily go back to the topic at a later date.



**Figure 1-7:** InfoCenter helps you to find out more about AutoCAD.

### Exercise 1-10: Searching for Information Using InfoCenter

Practice searching for information in AutoCAD using the InfoCenter.

1. Click in the Search field and enter **create 3D box**.
2. Press Enter to start the search.
3. Position the cursor over the Create a Solid Box entry and click the star to the right of the link.

The link is now marked as a favorite.

4. Click the Favorites button located on the right-side of the InfoCenter (refer to Figure 1-7) and click the link that you added.

AutoCAD's Online Help system launches and displays the topic.

5. Close the Online Help system and click the Communication Center.

Browse through the topics shown. (The information is pulled from a number of different resources.) You can add your favorite RSS feeds to InfoCenter, which AutoCAD searches when you use the Search field of InfoCenter.

6. Click the down arrow to the right of the InfoCenter Search button, and choose Search Settings from the shortcut menu.
7. In the InfoCenter Settings dialog box, select RSS Feeds from the left pane, and click New.
8. In the Add RSS Feed dialog box, enter **http://hyperpics.blogs.com/beyond\_the\_ui/index.rdf** in the Enter a Location text box and then click Add.

9. When the RSS Feed Confirmation message box is displayed, click OK.
10. Click OK to save the changes made in the InfoCenter Settings dialog box.

To see if the new RSS feed was added, click Communication Center in the InfoCenter. You should see the five most recent items in the RSS feed listed in the HyperPics: Beyond the UI section.

## *Turning to the World Wide Web*

The World Wide Web is a great resource for selling or buying things, but it also happens to be a great place to get help on AutoCAD. The Internet is bursting with sites that offer information on AutoCAD and help when you're stuck. Chances are good that you can use a popular search engine such as Google ([www.google.com](http://www.google.com)) or Yahoo! ([www.yahoo.com](http://www.yahoo.com)) to find a solution to a problem that you might be having with AutoCAD. If you can't find a solution to your problem using a search engine, you can use an online community site like AUGI ([www.augi.com](http://www.augi.com)) or the online discussion forums that Autodesk supports ([www.autodesk.com/discussion](http://www.autodesk.com/discussion)) to post your problem and wait for someone to respond with a solution. In addition to these Web sites, I list resources on the Internet in Chapter 23.

## *Answers to Practice Opportunities*

### **1. How can you start AutoCAD?**

A, B, D. You can launch AutoCAD with the following methods:

- Choose Start ⇨ (All) Programs ⇨ Autodesk ⇨ AutoCAD 2008 ⇨ AutoCAD 2008.
- Double-click an AutoCAD drawing file in any folder on a local or network drive.
- Double-click the AutoCAD icon on the Windows desktop.
- Click the AutoCAD icon on the taskbar's Start menu.
- Click the AutoCAD icon on the Quick Launch toolbar on the taskbar.

### **2. To access the shortcut properties in order to add a command line switch, right-click and choose Properties.**

### **3. Besides typing the command at the command prompt, how else can you start a command?**

A and C. You can select a menu item from a pull-down menu or shortcut menu, or you can click a button on a toolbar or the dashboard.

## Chapter 2

# Working with Drawing Files

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### *In This Chapter*

- ▶ Creating a new drawing
  - ▶ Opening an existing drawing
  - ▶ Navigating open drawing windows
  - ▶ Saving a drawing
  - ▶ Protecting your drawings using passwords and digital signatures
  - ▶ Closing a drawing
  - ▶ Managing drawing files
  - ▶ Archiving and backing up drawing files
  - ▶ Recovering a drawing file
- 

You do a lot of work with drawing files when using AutoCAD; the main byproduct of AutoCAD is a drawing file, which has the file extension DWG. The exercises in this chapter give you an understanding of how to create, open, save, manage, and navigate drawing files in AutoCAD.

## *Creating a New Drawing*

When you first start AutoCAD, it creates a default drawing automatically, but commonly you don't want to use this default drawing when creating a new drawing (unless you use a custom shortcut to specify a drawing template). To create a new drawing file, you use either the NEW or QNEW command.



The `/t` command line switch allows you to specify a drawing template that contains items such as layers, blocks, and other drawing-related options and objects that you want to have access to when creating a new drawing. Chapter 1 covers command line switches.

You can create a new drawing in the following ways:

- ✓ Choose File→New and select a drawing template, start from scratch, or use the wizard.
- ✓ Click Qnew on the Standard toolbar and select a drawing template if a default template isn't set up.
- ✓ Press Ctrl+N and select a drawing template, start from scratch, or use the wizard.

When you create a new drawing, you can use a drawing template which contains the pre-defined formats and settings you want to use, or you can create a drawing from scratch using all the default settings that are defined by AutoCAD for a drawing. By default, the NEW

command displays the Select Template dialog box, which allows you to create a drawing by using a template or starting from scratch, but if the STARTUP system variable is set to 1 instead of the default value 0, the Create New Drawing dialog box is displayed instead. The Create New Drawing dialog box allows you to create a new drawing by using a drawing template, starting from scratch, or using one of the wizard options.

## Drawing templates

AutoCAD ships with a number of drawing template files that you can choose from, or you can create your own template based on the settings that you want preset. A drawing template is just like a drawing file except it has the DWT file extension. When you use a template to create a new drawing, AutoCAD gives the new drawing a default name instead of opening an existing drawing and saving the file with a different name. The default name that's assigned to a drawing file when using a drawing template is Drawing1, Drawing2, and so on. By not opening and saving a drawing file with a different name, you avoid the issue of accidentally saving over the file.



Drawing template files are stored by default on your local drive, but if you're working in a large group of individuals who need to use the same set of drawing templates, you can tell AutoCAD to look for them on a network drive instead. To specify a new path for the drawing templates, choose Template Settings → Drawing Template File Location. The Options dialog box appears, and you can change the path on the Files tab.

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### Exercise 2-1: Creating a New Drawing from a Template

Practice creating a new drawing based on a drawing template file.

1. Choose File → New.

The Select Template dialog box lists the drawing template files that are indicated in the path set in the Options dialog box.

2. In the Select Template dialog box, select the drawing template file to use and click Open.

A new drawing named Drawing2 is created. Subsequent new drawings are named Drawing3, Drawing4, and so on.

3. Close the drawing without saving it.

Choose File → Close or the X button in the upper-right corner of the drawing window.



1. To specify a default drawing template when creating a new drawing, enter \_\_\_\_\_ at the command prompt to use the \_\_\_\_\_ command.

---

### Exercise 2-2: Creating a Custom Template

Practice creating a new custom drawing template.

1. Create a new drawing using the acad.dwt drawing template file.
2. Add two new layers to the drawing called Center and Dims with the colors 5 and 6, respectively.





To add a new layer, first choose Format→Layer to display the Layer Properties Manager. In the Layer Properties Manager, click New Layer and enter the name **Center**. Click in the Color column for the Center layer and change the color to color 5 on the ACI tab. Create the Dim layer and assign it the color 6 by using the same steps that you used to create the Center layer.



3. Add other objects such as blocks, dimension styles, or anything else that you might want to be part of the drawing template.

You can use the DesignCenter to add named objects from one drawing to another drawing that you have open. This can help you save time when creating a drawing template with the layers and styles that might already exist in another drawing.

4. Choose File→Save As.
5. In the Save Drawing As dialog box, select AutoCAD Drawing Template (\*.dwt) from the Files of Type drop-down list.
6. In the File Name text box, enter the name **MyTemplate**.
7. Under My Documents, open the My Practice Drawings folder and then browse to the Chapter 2 folder.
8. Click Save.
9. In the Template Description dialog box, highlight the text that's present and enter **My Custom Drawing Template**.
10. Select English from the Measurement drop-down list and click OK.
11. Close the current drawing.
12. Create a new drawing by using the My\_Template.dwt file.  
Display the Layer Properties dialog box, and the two layers that you added to the drawing template exist in the drawing.
13. Close the drawing without saving it.

## *Starting from scratch*

When creating a new drawing, you have the option to create a new drawing from scratch rather than basing it on a drawing template. The only options you have to choose from are the Imperial (inches, feet, and so on) and Metric (millimeter, meter, and so on) measurement options, which affect the linetypes and hatch patterns used for the drawing; otherwise, the drawing file is created with all the default drawing settings, and no named objects, such as layers or text styles, are created in the drawing.

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### **Exercise 2-3: Creating a Drawing from Scratch**

Practice creating a new drawing from scratch with the Imperial measurement setting.

1. Choose File→New.
2. In the Select Template dialog box, click the drop-down arrow next to the Open button and click Open with No Template - Imperial.

A new drawing is created with only the measurement setting set to Imperial and all other settings set to their default values.

3. Close the drawing without saving it.

## Calling upon the wizard

The wizard options that come with AutoCAD really aren't used for production drawings because they're rather limited in the settings that they can control, but they're nice for creating custom drawing templates because they help you set up some of the very basic drawing settings. Everything the wizard offers can be done manually with the help of a few commands that are in AutoCAD.

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### Exercise 2-4: Creating a Drawing with the Use a Wizard Option

Practice creating a new drawing using the Use a Wizard Option of the Create New Drawing dialog box.

1. At the command prompt, enter **startup** and press Enter.
2. At the prompt `Enter new value for STARTUP <0>:`, enter **1** and press Enter.
3. Choose File→New.
4. In the Create New Drawing dialog box, click Use a Wizard.
5. Select Advanced Wizard and click OK.

The Quick Wizard offers fewer options than the Advanced Wizard does, so be sure to select the Advanced Wizard.
6. Select Decimal for the unit of measurement and 0.000 from the Precision drop-down list. Click Next.
7. Select Decimal Degrees for the angle of measurement and 0.00 from the Precision drop-down list. Click Next.
8. Click Next to keep 0 degrees in the direction of East.
9. Click Next to keep positive angle entry going in the counterclockwise direction.
10. Enter **17** in the Width text box and **11** in the Length text box. Click Finish.

The Advanced Wizard sets the drawing and angular units, along with the Model space limits for the drawing. To set both the drawing and angular units of a drawing, use the UNITS command, and to set the drawing limits, use the LIMITS command.
11. Change the STARTUP system variable back to a value of 0 (zero) by completing Steps 1 and 2, and when prompted for a new value enter **0** instead of 1.
12. Close the drawing without saving it.

## Opening an Existing Drawing

After you create a drawing, it's only a matter of time until you need to reopen it to make some revisions or plot it off for someone to review. AutoCAD allows you to open drawing files (DWG), drawing template files (DWT), drawing standards files (DWS), and drawing exchange files (DXF). Choose File→Open and then browse to the file you want.



2. Besides choosing File⇨Open, how else can you open an existing drawing?
  - A. Choose File⇨Re-open and select the drawing file to open.
  - B. Choose a drawing from the most recent list at the bottom of the File menu.
  - C. Click Open on the Standard toolbar and select the drawing file to open.
  - D. Press Ctrl+O and select a drawing file to open.

### Exercise 2-5: Opening a Drawing File

For this exercise, open the drawing file `Exercise2-5.dwg` from the Chapter 2 folder in the My Practice Drawings folder under My Documents. Then practice opening an existing drawing file.

1. Choose File⇨Open.
2. In the Select File dialog box, select Drawing (\*.dwg) from the Files of Type drop-down list.

If you want to open a drawing standard, drawing template, or drawing exchange file, just select the appropriate option from the Files of Type drop-down list.

3. Browse to and select the drawing file `Exercise2-5.dwg` from the Chapter 2 folder in the My Practice Drawings folder under My Documents, and click Open.
4. Minimize AutoCAD by clicking the Minimize button located in the upper-right corner of the application window.
5. Use the AutoCAD 2008 shortcut on your desktop to start a second session of AutoCAD.
6. In the new session of AutoCAD, browse to and open the `Exercise2-5.dwg` file.
7. When prompted to open the file in read-only mode, click Yes.

The message box lists the name of the computer that the file is currently opened on. This can be helpful if you need to work on a drawing that someone else has open.

8. At the command prompt, enter **whohas** and press Enter.

WHOHAS allows you to retrieve who currently has a drawing file open and when it was opened. This can be helpful when working on a project that has multiple drafters assigned to it.

9. In the Select Drawing to Query dialog box, browse to and open the `Exercise2-5.dwg` file again.

A message box displays the path to the drawing, along with the computer that it is opened on and when it was opened.

10. Close both sessions of AutoCAD without saving the changes to the drawing.

If your drawing file contains named views, you can click the Select Initial View option to choose a named view when the drawing is being opened.

At times, you might want to open a drawing to only view or plot it. To avoid accidentally saving changes to the file, you can click the drop-down arrow next to the Open button and select Open Read-Only to mark the file as read-only. Using the Open Read-Only option doesn't affect the actual read-only attribute of the drawing file located on the hard drive or network drive.



## Navigating Drawing Windows

After you start creating and opening drawing files, chances are good that you'll quickly find out that AutoCAD allows you to have multiple drawing files open at a time. AutoCAD allows you to work with multiple drawing files just like most other Windows-based applications do. You can select one of the open drawings from the Window menu, arrange the drawings in the document area by tiling or cascading them, or press Ctrl+Tab and Ctrl+Shift+Tab to cycle through the open drawing files.



Having the ability to work with multiple drawings open at a time is great, but it does take some additional understanding of how to work efficiently with them, and it does take additional resources. Be sure you need more than one drawing open at a time.

---

### Exercise 2-6: Arranging Drawing Windows

Practice arranging multiple drawing windows.

1. Create two new drawings or open two existing drawings that you have already created.
2. Choose Window⇨Cascade.

The drawing windows are no longer maximized and are arranged starting in the upper-left corner of the document area stacked diagonally towards the lower-right corner. The drawing windows are arranged so you can see the title bars of each open drawing but not the drawing area of each one.

3. Choose Window⇨Tile Horizontally.

The drawing windows are now arranged so they're displayed one above the other in the document area. You can see the title bar and the drawing window of both the open drawings.

4. Choose Window⇨Tile Vertically.

The drawing windows are now arranged so they are displayed side by side in the document area. You can see the title bar and the drawing window of both the open drawings.

5. Close the drawings without saving them.

AutoCAD allows you to copy content between drawings by dragging from one drawing and dropping into another one. If you drag and drop with the left mouse button, you can copy and paste from one drawing to another. If you drag and drop between drawings with the right mouse button, you get options to perform a copy, paste as a block, or paste to original coordinates on the shortcut menu.



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### Exercise 2-7: Switching between Open Drawings

Practice switching between open drawings.

1. Create three new drawings; you don't need to save the drawings.
2. Watch the title bar of AutoCAD or the current drawing window, and press Ctrl+Tab.

If the drawing windows are maximized, the name of the drawing is changed in the title bar of AutoCAD and if the drawing windows are tiled, the focus of the current drawing window switches to the previous one that was active.

3. Press Ctrl+Shift+Tab, and watch the title bar of AutoCAD or the current drawing window.  
Ctrl+Shift+Tab is like Ctrl+Tab; the only difference is the direction that the open drawings are cycled in.
4. Choose Window⇨*drawing name* to activate a specific drawing file that's open.
5. Close the drawings without saving them.



If you have more than nine drawings open at a time, click the More Windows option at the bottom of the Window menu to display the Select Window dialog box, which allows you to see a list of all the drawings that you currently have open.

## Saving Drawings

After you've worked on your design for a little while, saving it is a good idea so you don't lose any of your work up to the current point. Saving a drawing in AutoCAD isn't much different from other Windows-based applications.

You can save drawings with these methods:

- ✓ Choose File⇨Save, and enter a name and path for the drawing if prompted for one.
- ✓ Click Save on the Standard toolbar.
- ✓ Enter **save** at the command prompt and enter a name and path for the drawing if prompted for one.
- ✓ Press Ctrl+S.



3. Name three ways to save a drawing with a different name.

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### Exercise 2-8: Saving a Drawing File

Practice opening an existing drawing.

1. Create a new drawing.
2. Choose File⇨Save.
3. In the Save Drawing As dialog box, enter the name **P04235-B** in the File Name text box and specify the location to save the drawing in.
4. Select AutoCAD 2007 Drawing (\*.dwg) from the Files of Type drop-down list.

If you need to save the drawing to an older format for someone else to open, select the appropriate option from the File of Type drop-down list.

5. Under My Documents, open the My Practice Drawings folder and then browse to the Chapter 2 folder.
6. Click Save.
7. Close the drawing.



You can change the default drawing format when saving to an older release if you need to. This allows you to not have to remember selecting a different format from the Files of Type drop-down list. To change the default file format when saving a drawing, select the format from the Save As drop-down list on the Open and Save tab of the Options dialog box (choose Tools⇨Options).

After you save a drawing more than once, AutoCAD renames the previous drawing file as a backup drawing with the file extension BAK. The backup drawing file allows you to recover the previous version of the drawing file that was saved. Backup drawing files come in handy if AutoCAD unexpectedly crashes due to an error in the application or if your computer crashes.

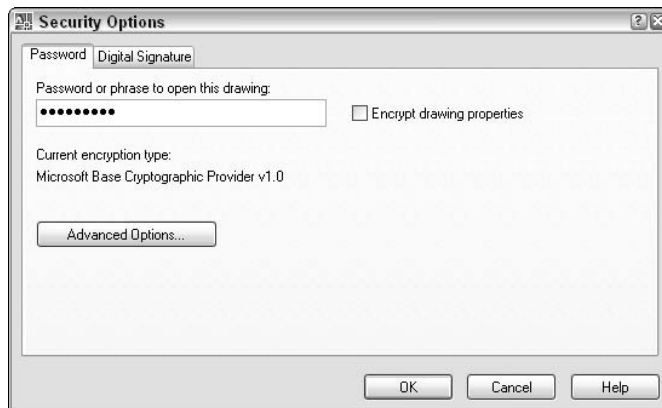


You should save your drawings as often as you can to eliminate any work that might be lost if your computer loses power or AutoCAD crashes unexpectedly. As an additional safeguard against losing your work, AutoCAD offers a feature called AutoSave, which performs a save after a specific time interval. You can customize the time interval on the Open and Save tab of the Options dialog box (choose Tools⇨Options) under the File Safety Precautions section. When the time interval is reached, AutoCAD saves a file with the extension SV\$ to the path specified under the Temporary Drawing File Location specified on the Files tab of the Options dialog box. A drawing autosave (SV\$) file is a drawing file, just with a different extension; when you change the SV\$ extension to DWG, you can open the file in AutoCAD again.

## Password-protecting your documents

If you find yourself sharing files with others outside of your office, you most likely want to protect your drawings from unwanted eyes when sending drawing files via e-mail or across the Internet. AutoCAD allows you to password-protect your drawings, which forces anyone who wants to view the drawings to enter a password that you supply. The Security Options dialog box (see Figure 2-1) allows you to assign a password or attach a digital signature to a drawing.

**Figure 2-1:**  
You use the Security Options dialog box to password-protect and digitally sign a drawing.



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**Exercise 2-9: Password-Protecting a Drawing File**

Practice saving a drawing with a password.

1. Create a new drawing.
2. Choose File⇨Save As.
3. In the Save Drawing As dialog box, choose Tools⇨Security Options.
4. In the Security Options dialog box, click the Password tab.
5. In the Password or Phrase to Open This Drawing text box, enter **protect30** or another password of your choice.
6. Click Encrypt Drawing Properties to hide the custom properties of a drawing when they're viewed outside AutoCAD.  
Encrypting the properties of a drawing is optional.
7. Click Advanced Options to specify an encryption provider and key length. Select *RC4, Microsoft Strong Cryptographic Provider* from the list of choices in the Choose an Encryption Provider list box.  
The encryption provider and key length can vary from computer to computer and country to country, and changing the values is optional.
8. Select 128 from the Choose a Key Length drop-down list and click OK.
9. In the Security Options dialog box, click OK.
10. In the Confirm Password dialog box, enter the password that you entered in Step 5 in the text box and click OK.
11. Finish saving the drawing with the name **ProtectedDWG**. Save it to the Chapter 2 folder in the My Practice Drawings folder under My Documents.
12. Close the drawing.



When saving a drawing with a password, be sure to write down the password that you use in a safe spot because there is no way to obtain the password if you lose it. You might also want to save the drawing with a password to a different location so you still have the original drawing file just in case you forget the password.

---

**Exercise 2-10: Opening a Drawing File That Is Protected by a Password**

For this exercise, open the drawing `ProtectedDWG.dwg` that you created in Exercise 2-9 or the drawing file `Exercise2-10.dwg` from the Chapter 2 folder in the My Practice Drawings folder under My Documents. Practice opening a drawing that has been previously password-protected.

1. In the Password dialog box, enter the password **protect30** or the one that you chose to use when saving the drawing.
2. Click OK to open the drawing.

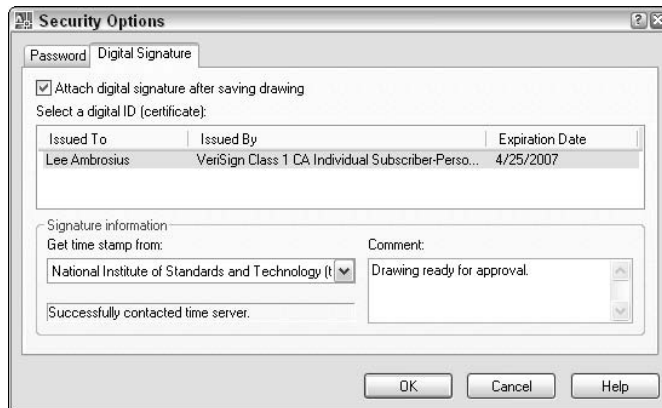
If you reference or insert a drawing that's protected by a password, you also need to enter the password when performing these tasks as well. After you open, attach, or insert a drawing with a password, you don't need to re-enter the password until you close and restart AutoCAD because it gets cached for the current session.

3. Close the drawing without saving changes.

## Attaching a digital signature

Digital signatures allow you to protect drawing files without restricting access to them like password protection does. When you digitally sign a drawing file, you are putting a stamp on the file that tells the next person that opens the drawing that it hasn't been altered since the last time it was saved. Unlike password-protecting a drawing (which is built into AutoCAD), you must obtain a digital signature from a provider such as VeriSign ([www.verisign.com](http://www.verisign.com)) or IdenTrust ([www.identrust.com](http://www.identrust.com)) before you can digitally sign a drawing.

VeriSign is the preferred choice of Autodesk because when no digital signature is found on your machine, you're sent to the Web site for VeriSign. After you obtain a digital signature, you use the Digital Signature tab of the Security Options dialog box (see Figure 2-2). If you save a drawing that's digitally signed on a different computer from the one you originally used to sign the drawing, the digital signature becomes invalid unless you resign the drawing.



**Figure 2-2:**  
The Digital  
Signature  
tab of the  
Security  
Options  
dialog box.

### Exercise 2-11: Obtaining a Digital Signature

Practice obtaining a trial digital signature from VeriSign.

1. Choose Tools⇨Options.
2. In the Options dialog box, click the Open and Save tab.
3. Click the Security Options button.
4. In the Security Options dialog box, click the Digital Signature tab.
5. In the Valid Digital ID Not Available dialog box, click Get a Digital ID.

The Products & Services page on VeriSign.com is displayed ([www.verisign.com/products-services](http://www.verisign.com/products-services)). If the Valid Digital ID Not Available dialog box isn't displayed, you most likely already have a valid digital signature present on your machine.

6. On the Products & Services page, click the Managed PKI Services link.



7. On the Managed PKI Services page, click the PKI Applications link under the Managed PKI Services Include section.
8. On the PKI Applications page, click the Digital IDs for Secure Email link.
9. On the Digital IDs for Secure Email page, click Buy Now.
10. On the Digital ID Enrollment page, click Buy Now again.
11. On the Choose Your Browser page, click Microsoft or Netscape (Netscape, Mozilla, and Apple Safari).
12. On the Step 1 of 4: Complete Enrollment Form page, enter the required information and click Accept.

After the information is submitted, you receive an e-mail with additional instructions on how to finish obtaining the digital signature and install it. Follow the instructions to complete the remainder of the process of obtaining and installing the digital signature.

---

### Exercise 2-12: Attaching a Digital Signature to a Drawing File

Practice attaching a digital signature to a drawing.

1. Create a new drawing.
2. Choose File→Save As.
3. In the Save Drawing As dialog box, choose Tools→Security Options.
4. In the Security Options dialog box, click the Digital Signature tab.
5. Click the Attach Digital Signature After Saving Drawing check box.  
If the Valid Digital ID Not Available dialog box is displayed, you need to obtain and install a digital signature from VeriSign (refer to Exercise 2-11).
6. Select the digital signature you want to attach to the current drawing.
7. Select National Institute of Standards and Technology from the Get Time Stamp From drop-down list.  
Specifying a computer or server from which to obtain the time stamp is optional.
8. Enter **Drawing ready for approval.** in the Comment text box.  
Specifying a comment when digitally signing a drawing is optional.
9. Click OK.
10. Finish saving the drawing with the name **SignedDWG**. Save it to the Chapter 2 folder in the My Practice Drawings folder under My Documents.
11. If the Signing Data with Your Private Exchange Key dialog box is displayed, click OK.
12. Close the drawing.



If you have a number of drawings that need to be digitally signed, you can use the Attach Digital Signatures utility that comes with AutoCAD. The only difference between attaching a digital signature in AutoCAD or using the Attach Digital Signatures is that you need to specify which drawings you want to digitally sign; otherwise, the process is the same. To launch the Attach Digital Signatures utility, choose Start→[All] Programs→Autodesk→AutoCAD 2008→Attach Digital Signatures.

### Exercise 2-13: Opening a Drawing with a Digital Signature

For this exercise, open the drawing `SignedDWG.dwg` that you created in Exercise 2-12 or open the drawing file `Exercise2-13.dwg` from the Chapter 2 folder in the My Practice Drawings folder under My Documents. Practice opening a drawing that has been previously digitally signed.

1. In the Digital Signature Contents dialog box, review the digital signature to make sure that the digital signature is valid.

The Digital Signature Contents dialog box open automatically when you open a drawing with a digital signature. If it doesn't, enter **sigvalidate** at the command prompt and press Enter to display the Validate Digital Signature dialog box. In the Validate Digital Signature dialog box, click the View Base Signature button to display the Digital Signature Contents dialog box.

2. Click Close.

When a drawing has a digital signature attached to it, the Validate Digital Signature icon is displayed in the status bar tray.

3. Close the drawing without saving changes.

If you make changes to a drawing file that has a digital signature attached to it, the digital signature becomes invalid unless you resign the drawing.



## Closing Drawings

After you save a drawing or if you don't want to save any of the changes you made, you can close the drawing file. AutoCAD allows you to close one drawing at a time or all the open drawings. When closing a drawing, AutoCAD checks to see whether the drawing has been saved, and if it hasn't, you're prompted to save or discard the changes or to cancel the closing of the drawing altogether.

You can close the open drawing in the following ways:

- ✓ Choose File→Close and save the drawing if prompted.
- ✓ Choose Window→Close and save the drawing if prompted.
- ✓ Enter **close** at the command prompt and save the drawing if prompted.
- ✓ Click the X on the drawing window.



4. How do you close all open drawings at the same time?
  - A. Choose File→Exit and save each drawing that needs to be saved.
  - B. Enter **closedrawings** at the command prompt and save each drawing that needs to be saved.
  - C. Choose Window→Close All and save each drawing that needs to be saved.
  - D. Enter **closeall** at the command prompt and save each drawing that needs to be saved.
  - E. Click the Close button on the application window and save each drawing that needs to be saved.

---

**Exercise 2-14: Closing a Drawing**

Practice closing a drawing.

1. Create a new drawing or open a drawing from the Chapter 2 folder in the My Practice Drawings folder under My Documents.
2. Choose Window→Close.
3. If an AutoCAD message box appears, asking you to save the changes, click No to discard the changes. If you want to save the changes, click Yes or click Cancel to keep the drawing open and not save the drawing.

---

**Exercise 2-15: Closing All Open Drawings**

Practice closing all the open drawing files.

1. Create a new drawing and open a few of the drawings from the Chapter 2 folder in the My Practice Drawings folder under My Documents.
2. Choose Window→Close All.
3. If an AutoCAD message box appears, asking you to save the changes, click No to discard the changes. If you want to save the changes, click Yes or click Cancel to keep the drawing open and terminate the closing of all the remaining open drawings.
4. Continue to click Yes or No when prompted to save changes for each drawing until all drawing files have been closed.

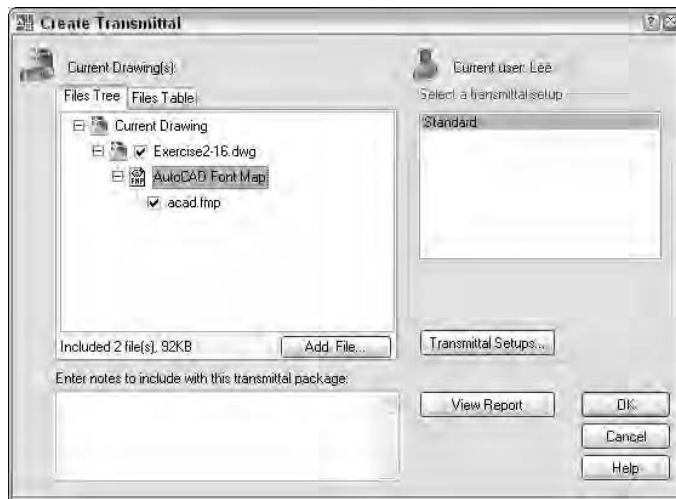


If you're done with AutoCAD for the day, you can close the application window. You're prompted to save any changes that you made to any of the drawings that are currently open before AutoCAD closes.

## *Archiving and Backing Up Drawings*

You spend a lot of time working on your drawings, so you should make sure to protect them from hardware failures through archives and backups. You shouldn't store drawing files locally on your computer, but on a network where your files can be backed up periodically and accessed by other members of the design team. If you don't have a network available to you, use a drive that can read/write to a removable media such as a DVD/CD or removable hard drive. Before you decide to create your own backups, check with your IT department or the CAD manager/lead drafter to see whether you need to follow an established backup process.

Archiving a drawing is a little more involved than just backing up a drawing file. Archiving a drawing file is like taking a snapshot in time of the drawing and all the files that the drawing relies on to be displayed and plotted correctly. AutoCAD offers two commands that allow you to archive drawings and all of its associated files. The ETRANSMIT command displays the Create Transmittal dialog box (see Figure 2-3), which allows you to create a transmittal package for a drawing with all of its associated files into an archive (ZIP or EXE) file or a folder. The other command that AutoCAD offers to archive a drawing is called ARCHIVE, which is useful if you're archiving sheet sets instead of individual drawings.



**Figure 2-3:**  
The Create  
Transmittal  
dialog box.

### Exercise 2-16: Creating a Transmittal Package of Your Drawings

For this exercise, open the drawing file `Exercise2-16.dwg` from the Chapter 2 folder in the My Practice Drawings folder under My Documents. Then practice packaging a drawing with the `ETRANSMIT` command and all of its associated files to a ZIP file.

1. Choose `File`→`eTransmit`.
2. In the Create Transmittal dialog box, click `Transmittal Setups`.
3. In the Transmittal Setups dialog box, click `New`.
4. In the New Transmittal Setup dialog box, enter **MyTransmittal** in the New Transmittal Setup Name text box.
5. Click `Continue`.
6. In the Modify Transmittal Setup dialog box, select `Zip (*.zip)` from the Transmittal Package Type drop-down list.
7. Click `Browse` under Transmittal File Folder and select the My Documents folder.
8. Select the `Prompt for a Filename` option from the Transmittal File Name drop-down list.
9. In the Transmittal Setup Description text box, enter **My Custom Transmittal Setup**.
10. Click `OK`.
11. In the Transmittal Setups dialog box, click `Close`.
12. In the Create Transmittal dialog box, click the `Files Table` tab to see which files will be included in the ZIP file that is created.
13. Click `OK`.
14. In the Specify Zip File dialog box, enter **MyTransmittal** in the File Name text box and browse to the Chapter 2 folder in the My Practice Drawings folder under My Documents.

### 15. Click Save.

The file `MyTransmittal.zip` is created in the Chapter 2 folder. The ZIP file that's created contains the files that were displayed on the Files Table tab.

## Recovering a Drawing

Computers are more and more like people each day, which is kind of scary. In this day and age, computers find themselves catching colds and overexerting themselves too much at times, which leads to something breaking or even worse — a system failure. Just like you take care of yourself by eating right, taking vitamins, and exercising, you should make sure to use virus scan software on your computer and network, and perform routine maintenance to keep them running smoothly. No matter how well you take care of yourself and your computers, something is bound to happen once in a while that is out of your control.

From time to time, you'll encounter a bad drawing file. Bad drawing files occur due to an internal problem in AutoCAD or a bad third-party application (an application that writes to the drawing [DWG] file format that isn't created by Autodesk), or a system crash occurs when AutoCAD is attempting to write to a drawing file. In these situations, AutoCAD provides some utilities that help you recover and audit drawing files that go bad. To recover a drawing file that you can't open in AutoCAD, you can use the RECOVER command. If a drawing has some issues but you can open it, you can use the AUDIT command.

If AutoCAD crashes with a drawing file open that you haven't saved yet, you're prompted to save the file with `_recover` appended to the end of the drawing's name. After restarting AutoCAD, the Drawing Recovery Manager palette (see Figure 2-4) displays, which allows you to open the drawing file that was open before the crash, the most recent backup drawing (BAK), automatic save (SV\$) files, and the drawing that AutoCAD saved for you before it was completely closed. The Drawing Recovery Manager palette displays information about the files that can be recovered and a preview of the last saved view in the file.

**Figure 2-4:** The Drawing Recovery Manager palette allows you to open files related to a drawing after a recent crash.



### Exercise 2-17: Recovering a Drawing

In this exercise, you use the drawing file `Exercise2-17.dwg` from the Chapter 2 folder in the My Practice Drawings folder under My Documents. Practice recovering the drawing file.

1. Choose `File`→`Drawing Utilities`→`Recover`.
2. In the Select File dialog box, browse to and select the `Exercise2-17.dwg`. Click `Open`.

The drawing file is opened in memory, and AutoCAD performs an audit on it before loading it into a drawing window. The results of the audit are displayed in the Text Window. The following is an example of the results an audit performed on a drawing file:

```
Auditing Header
Auditing Tables
Auditing Entities Pass 1
Pass 1 500      objects audited
Auditing Entities Pass 2
AcDbLayerTable(2)
  Extension dictionary 761 Not this object      Set to this
AcDbLayerTableRecord: "0"
  Extension dictionary 81E Not this object      Set to this
AcDbLayout(5E)
  could not be repaired. It
  will be Erased.
AcDbTableStyle(87)
  Extension dictionary 826 Not this object      Set to this
AcDbLayerTableRecord: "CENTER"
  Extension dictionary 824 Not this object      Set to this
Pass 2 500      objects audited
Auditing Blocks
1      Blocks audited
Total errors found 5 fixed 5
Erased 1 objects
```



To display the Text Window, press `F2`.

You can use the `RECOVERALL` command (choose `File`→`Drawing Utilities`→`Recover Drawing and Xrefs`) to recover any drawing files that are attached to the drawing file that you select to recover.

### Exercise 2-18: Auditing a Drawing

Open the drawing file `Exercise2-18.dwg` from the Chapter 2 folder in the My Practice Drawings folder under My Documents and then practice auditing the drawing file.

1. Choose `File`→`Drawing Utilities`→`Audit`.
2. At the prompt `Fix any errors detected? [Yes/No] <N>:`, enter `Y` and press `Enter`.

AutoCAD performs an audit on the drawing file, and the results are displayed in the Text Window.

---

**Exercise 2-19: Using the Recovery Manager**

Open the drawing file `Exercise2-19.dwg` from the Chapter 2 folder in the My Practice Drawings folder under My Documents. Practice using the Recovery Manager to open a drawing file after AutoCAD crashes.

1. Save the drawing and check the Chapter 2 folder. A backup drawing file is now created for the drawing
2. Choose Tools⇨Options.
3. In the Options dialog box, click the Open and Save tab.
4. Under the File Safety Precautions section, change the current value for Minutes Between Saves to 1.

Take note of the current value so you can change it back later; the default value is 10.

5. Click OK.
6. Choose Draw⇨Point⇨Single Point and click anywhere in the drawing.

No need to worry about what the Point command does, but just in case you do want to know it creates a point object at the point you picked. The AutoSave feature only begins after a change has occurred in a drawing; saving a drawing results in the AutoSave feature being temporarily disabled. Wait a minute until a message similar to *Automatic save to C:\DOCUME~1\Lee\LOCALS-1\Temp\Exercise2-19\_1\_1\_6970.svg* is displayed in the Command Line window.

When you have a drawing, a backup drawing, and an automatic save file to work with, you have to make AutoCAD think it's crashing so you can see how the Recovery Manager works.

7. Right-click an empty area of the Windows taskbar and choose Task Manager.  
Or choose Start (Windows)⇨Run. In the Run dialog box, enter **taskmgr** in the Open text box and click OK.
8. In the Windows Task Manager, click the AutoCAD 2008 entry on the Applications tab.
9. Click the End Task button.
10. Wait until the End Program dialog box displays and click the End Now button.
11. If you're prompted to save the recent changes, click OK.

The changes aren't saved to the original file; instead, they're saved to a drawing file of the same name as the drawing file that was open during the crash, but with `_recover` appended to the end of the filename.

12. Restart AutoCAD.
13. In the Drawing Recovery dialog box, click OK.  
The Drawing Recovery dialog box is letting you know that something caused AutoCAD to close unexpectedly.
14. On the Drawing Recovery Manager palette, click the plus sign next to the Exercise2-19 folder in the Backup Files section.

All the files that contain the most recent changes of the drawing that was open during the crash are displayed in this folder. The original `Exercise2-19.dwg` drawing is listed, along with an automatic save file and the most recent backup drawing file. If AutoCAD allowed you to save the drawing during the crash, you have a drawing file with the name `Exercise2-19_recover.dwg` in the list as well.

15. Click each drawing file to display a preview and details about the file.
16. Double-click the file `Exercise2-19.bak` to open the backup drawing file.
17. Save the drawing file with the name `Exercise2-19b`.  
Choose `File` → `Save As` to save the current drawing file with a different name.
18. Right-click the `Exercise2-19` folder in the Drawing Recovery Manager and select `Remove`.



If you save one of the files that you open from the Drawing Recovery Manager with the same name as the original drawing file, the other files are automatically removed along with the folder used to organize the files.

## Answers to Practice Opportunities

1. To specify a default drawing template when creating a new drawing, enter `qnew` at the command prompt to use the `QUICK NEW` command.

You specify the path and filename for the drawing template to be used by choosing `Template Settings` → `Default Template File Name for Qnew`. The Options dialog box appears, and you can specify the path and filename on the Files tab.

2. Besides choosing `File` → `Open`, how else can you open an existing drawing?

B, C, and D. You can open a drawing by selecting a file from the bottom of the File menu, clicking `Open` on the Standard or Standard Annotation toolbar, or by pressing `Ctrl+O`.

3. Name three ways to save a drawing with a different name.

Do any of the following and then enter a name and path:

- Choose `File` → `Save As`.
- Enter `saveas` at the command prompt.
- Press `Ctrl+Shift+S`.

4. How do you close all the open windows at once?

A, C, D, E. You can close all open drawings by choosing `File` → `Exit` or `Window` → `Close All`, entering `closeall` at the command prompt, or clicking the `Close` button on each application window. You're prompted to save any drawings you haven't saved.



## Chapter 3

# Customizing Your Work Environment

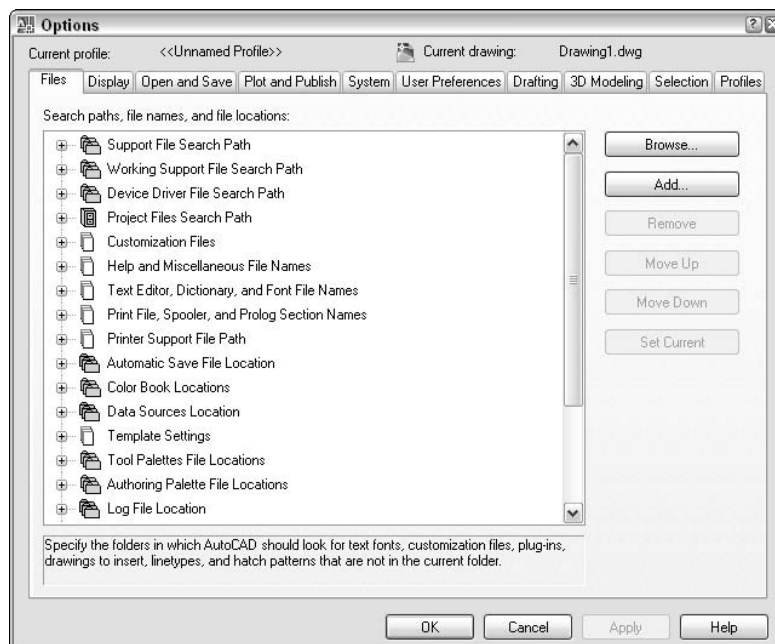
### *In This Chapter*

- ▶ Setting drafting options and creating user profiles
- ▶ Using and creating workspaces
- ▶ Customizing the user interface

AutoCAD offers a number of ways to make sure that the drawings you create are consistent with each other and allow you to access the right tools quickly when you need them. Some of the most basic ways of customizing AutoCAD and the user interface are the most effective in helping you to save time when creating and modifying drawings. The exercises in this chapter give you an understanding of how to do some basic customization to the drawing environment in AutoCAD and the user interface.

## *Setting Up Drafting Options*

AutoCAD uses a number of different options to control the behavior of the way objects are created and modified, how its application window looks, and even where it looks for files that are specific to some features. You can find most — but not all — of the options that AutoCAD uses in the Options dialog box (see Figure 3-1).



**Figure 3-1:**  
The Options  
dialog box.

You can open the Options dialog box with any of the following methods:

- ✓ Choose Tools⇄Options.
- ✓ Right-click the drawing window with no objects selected and choose Options.
- ✓ Right-click the Command Line window and choose Options.
- ✓ Enter **options** or **op** at the command prompt.

You can also access some settings when using a command such as FILLET and CHAMFER.



1. What is the Options dialog box used for?

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The Options dialog box is organized into ten tabs with options that are related to each other on each tab. For example, all the options relating to selecting objects are on the Selection tab, whereas the options related to working with objects in 3D are on the 3D Modeling tab. Each tab has a theme, but not all the options that affect a specific drafting task are located on just one tab. For example, when working with lights in your 3D model, you have options on the Files tab for where the photometric Web files are stored and more options on the Drafting tab for settings for the glyphs that control how lights look. It's really important to understand where the options are stored for the features that you use on a regular basis because you might find it possible to set an option once and then forget about it until you need to change it.

Table 3-1 lists the different tabs in the Options dialog box and the types of options on each one.

<i>Tab Name</i>	<i>Description</i>
Files	Contains the locations in which specific files are stored that AutoCAD uses for a number of different features. A number of file and folder locations affect working with 3D features, such as the location of texture maps and tool palettes.
Display	Contains settings that affect the way the application and the drawing window look. Settings on this tab affect the way the drawing window looks when working in a 2D or 3D view that uses either parallel or perspective projection.
Open and Save	Contains settings that affect opening and saving drawing files.
Plot and Publish	Contains settings that affect the way drawings are output to both electronic files and hardcopy.
System	Contains settings that affect the performance of AutoCAD at the application level. The main option on this tab that affects 3D is the Performance Settings button (under the 3D Performance section), which allows you to configure hardware acceleration.

<b>Tab Name</b>	<b>Description</b>
User Preferences	Contains settings that affect the way you interact with AutoCAD and specific features.
Drafting	Contains settings that affect selecting points in the drawing window and some settings that are related to working with 3D objects and visualization.
3D Modeling	Contains settings that affect the creation of 3D objects and provide input when working in the third dimension.
Selection	Contains settings that affect selecting objects in the drawing window.
Profiles	Contains controls to allow you to create and manage user profiles.

The remainder of this section provides an overview of the settings in the Options dialog box that affect 3D modeling in AutoCAD. To find out more about the settings in the Options dialog box, click the Help button in the Options dialog box to open the Online Help system or click the ? icon in the dialog box and click a control to display a tooltip with a description of the tool.

## ***Files tab***

The Files tab contains a number of settings that are related to the locations of files that AutoCAD uses, such as drawing templates, support files, and even the help files. Usually you don't need to change many of the default locations that AutoCAD uses, but at times you need to know where some of the files that you use with 3D features are stored so you can add more files to those locations.

### ***Tool Palettes File Locations***

The location(s) specified under Tool Palettes File Location affect the tool palettes that are displayed on the Tool Palettes Window, which includes the tool palettes associated with the Materials Library and other tool palettes that contain visual styles, materials, and lights.

### ***Texture Maps Search Path***

Specifies where AutoCAD looks for image files that are used for materials.

### ***Web File Search Path***

Specifies where AutoCAD looks for photometric Web files used with Web lights.

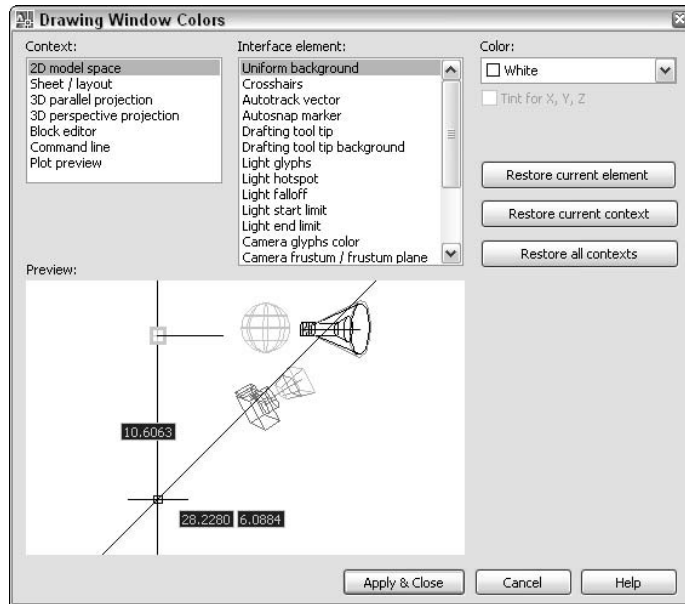
## ***Display tab***

The Display tab contains a variety of settings that affect the way the application and drawing windows look, along with how objects appear in the drawing. Features on this tab affect both how the drawing window looks when working in a 3D projected view and how some 3D helper objects look in the drawing window.

### ***Colors***

The Drawing Window Colors dialog box (see Figure 3-2) allows you to control the way the drawing window looks when working in one of the different contexts that are in AutoCAD. Two of the available contexts are used when the model in the drawing window

is displayed in a parallel or perspective projected view. To display the Drawing Window Colors dialog box, click the Colors button under the Window Elements section.



**Figure 3-2:**  
The  
Drawing  
Window  
Colors  
dialog box.

### ***Rendered Object Smoothness***

Specifies the smoothness of the curved surfaces of 3D objects when rendered or shaded.

### ***Contour Lines Per Surface***

Specifies the number of contour lines per surface that affects both the performance of a rendering and how long it takes to update the display of the drawing window.

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## **Exercise 3-1: Changing the Display of Interface Elements**

For this exercise, open a drawing. Then practice changing the appearance of the light glyphs in a drawing window for the 3D parallel projection context.

1. Choose Tools⇄Options.
2. In the Options dialog box, click the Display tab.
3. On the Display tab, click Colors.
4. In the Drawing Window Colors dialog box, click the 3D Parallel Projection from the Context list.
5. Select Light Glyphs from the Interface Element list.
6. Select Yellow from the Color drop-down list.
7. Click Apply & Close to save the changes.
8. Click OK to exit the Options dialog box.

## System tab

The System tab contains an odd assortment of settings that help to address application performance issues, from caching layout tabs to whether AutoCAD beeps if you input an incorrect value. In the odds and ends settings of the System tab is a lone button that allows you to configure hardware performance settings that AutoCAD uses. To access the hardware performance settings for AutoCAD, click the Performance Settings button under the 3D Performance section.

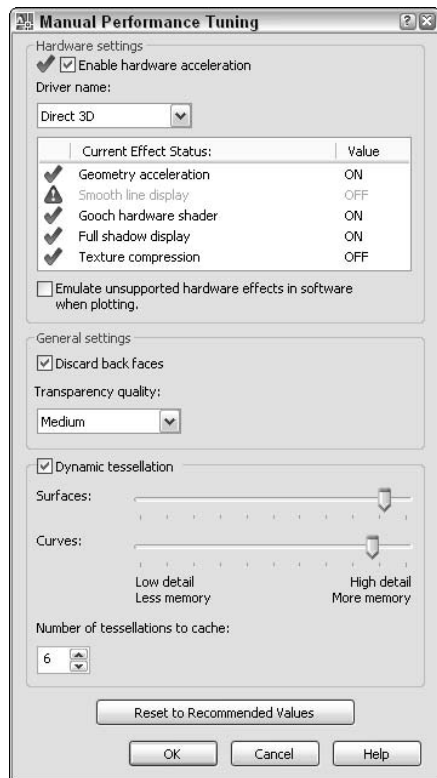
The display in AutoCAD is drawn one of two ways: by the software or hardware. I'm sure you're sitting there thinking, "Doesn't the software tell the computer how to draw the objects, and then doesn't the hardware display it?" Although the software does tell the hardware what to draw, when software acceleration is enabled, the CPU (central processing unit) is given the task of doing all the calculations to generate the image that's displayed on-screen. Hardware acceleration, on the other hand, shifts most calculations to the GPU (graphics processing unit) on the graphics card, and then an image is displayed on-screen.

Depending on the age of your computer's graphic card, it might not be certified for use with AutoCAD. Having a certified graphics card indicates that Autodesk has tested the card with AutoCAD, and you shouldn't have any types of display problems when using hardware acceleration. However, lacking a certified graphics card doesn't mean you can't run AutoCAD on your system; you just might encounter some issues with the display not refreshing properly at times, or at worst, a system crash might occur. If you're running a graphics card that isn't certified, you can check Autodesk's Web site ([www.autodesk.com/autocad-graphicscard](http://www.autodesk.com/autocad-graphicscard)) to see whether your graphics card has a newer driver that you can download and run so you can use all the features that your graphics card supports or whether you need to enable software acceleration.

The first time you start AutoCAD, it checks your graphics card against the database of graphics cards that were tested when it shipped. After the initial setup, you can use the Adaptive Degradation and Performance Tuning dialog box (see Figure 3-3) to change between software and hardware acceleration. Then you can specify which features of hardware acceleration to use from the Manual Performance Tuning dialog box (see Figure 3-4). Along with enabling and disabling hardware acceleration from the Adaptive Degradation and Performance Tuning dialog box, you can get information about your graphics card and change the settings of which effects shouldn't be degraded when your hardware can't support them as AutoCAD attempts to update the display in real time.

**Figure 3-3:**  
The  
Adaptive  
Degradation  
and  
Performance  
Tuning  
dialog box.





**Figure 3-4:**  
The Manual  
Performance  
Tuning  
dialog box.

### Exercise 3-2: Enabling Hardware Acceleration and Adjusting Adaptive Degradation

For this exercise, open a drawing. Then practice enabling hardware acceleration and changing the behavior that AutoCAD uses to degrade the effects that it supports when viewing a drawing in a shaded view in real time.

1. Choose Tools⇨Options.
2. In the Options dialog box, click the System tab.
3. On the System tab, click Performance Settings.
4. In the Adaptive Degradation and Performance Tuning dialog box, click Manual Tune to open the Manual Performance Tuning dialog box.
5. If the Enable Hardware Acceleration option is deselected, select it.

The Driver Name and Effects area becomes enabled. If an effect is unsupported, a red caution icon appears to the left of the effect's name, but if the effect has some sort of limitation, a yellow caution icon is displayed instead. When the feature is fully supported and there were no issues found during the certification of the driver by Autodesk, a green check mark is displayed next to the effect's name.

6. From the Driver Name drop-down list, select OpenGL. (If your hardware doesn't support it, select Direct 3D.)

Some graphics cards support custom drivers from the vendor, so you might want to check whether the manufacturer of your graphics card offers such a driver. A custom driver can make better use of the specific hardware capabilities of the hardware on your system, essentially getting more for what you paid for.

7. In the Hardware Effects list, select Full Shadow Display and then select ON from the drop-down list that's displayed.

Not all graphics cards support Full Shadow Display, but it's nice if you can enable the feature so you can see shadows in real time in the drawing window when user-defined lights are in use.

8. Click OK to return to the Adaptive Degradation and Performance Tuning dialog box.
9. In the Adaptive Degradation and Performance Tuning dialog box, make sure Adaptive Degradation is selected.

The Adaptive Degradation option is used when using commands such as 3DZOOM and 3DORBIT in a shaded viewport. As you zoom, pan, or orbit in a viewport, AutoCAD attempts to keep up with refreshing the screen as you change the view, but based on your hardware, it might not be able to keep the display clean-looking with all the effects that it supports. So as you change the view, the options in the Degradation Order list control the order in which effects should not be used if performance drops too much.

10. In the Degradation Order list, deselect Intersection Edges and Wireframe Display.

By deselecting the options, AutoCAD won't degrade the effects no matter how slow the display gets.

11. Select Textures from the Degradation Order list and move it above Ground Shadows by clicking the Move Up button as many times as it takes.

Textures now are degraded much earlier than they originally were. The order of the effects is personal; AutoCAD doesn't care which one it stops using first — AutoCAD just wants to go fast.

12. Click OK twice to save the changes you made.

## *Drafting tab*

The Drafting tab contains a number of settings that affect how you select points in the drawing window, how dynamic tooltips are used, and the appearance of the glyphs for lights and cameras.

### *Replace Z Value with Current Elevation*

Controls whether the current value of the ELEVATION system variable is substituted for the Z value of the coordinate that you pick in the drawing window when using object snaps.

### *Ignore Negative Z Object Snaps for Dynamic UCS*

Controls whether negative Z values are ignored when using Dynamic UCS with object snaps.

### Lights Glyph Settings

Displays the Light Glyph Appearance dialog box, which allows you to control the size and color used to display light glyphs in the drawing window. To display the Light Glyph Appearance dialog box, click the Lights Glyph Settings button on the Drafting tab.

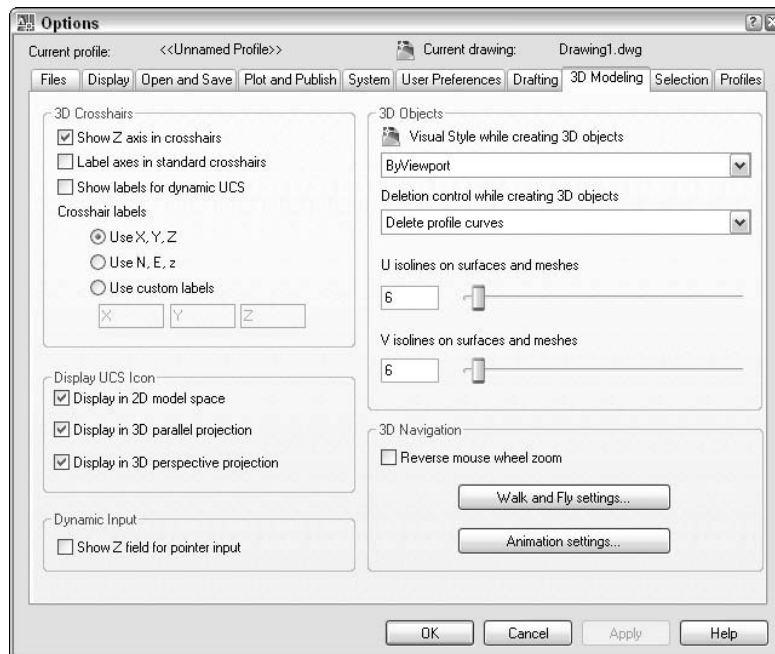
### Camera Glyph Settings

Displays the Camera Glyph Appearance dialog box, which allows you to control the size and color used to display camera glyphs in the drawing window. To display the Camera Glyph Appearance dialog box, click the Cameras Glyph Settings button on the Drafting tab.

## 3D Modeling tab

The 3D Modeling tab (see Figure 3-5) controls a lot of different settings used for creating and viewing drawings with 3D objects in them. Some of the settings affect the way the crosshairs are displayed in a 3D parallel or perspective projected view and how the UCS and dynamic input behave when working with 3D objects. Some settings also control 3D viewing commands such as 3DWALK and 3DFLY.

**Figure 3-5:**  
The 3D Modeling tab contains a number of settings that affect 3D drafting.



### 3D Crosshairs

The 3D Crosshairs section contains various settings that control how the crosshairs appear when working in a 3D projected view of the model.

### Display UCS Icon

The Display UCS Icon section contains settings that control in which context the UCS icon is displayed for both 2D and 3D.



### Dynamic Input

The Dynamic Input section contains an option that controls whether a field for Z coordinates is displayed when prompted for coordinate input or feedback of the current position of the crosshairs in the drawing when no command is active.

### 3D Objects

The 3D Objects section contains settings that affect the drawing window when creating 3D objects and how the final object look when created.

### 3D Navigation

The 3D Navigation section contains settings that affect the behavior of the scroll wheel on an IntelliMouse and how the 3DWALK and 3DFLY commands behave when used.

## Profiles tab

The Profiles tab allows you to create and manage user profiles. A *user profile* is a way to group and organize settings based on the type of task that you're doing or even project-based settings. Sometimes you want the display and settings to be a specific way when working on 2D models and then another way when working on 3D models. Profiles allow you to do just that without manually switching settings. Any setting in the Options dialog box without a drawing icon next to it is stored in the user profile. You can export and re-import user profiles as a way to back up your custom settings, and you can quickly configure other machines using the exported user profile. User profiles are exported to a file with the extension ARG.



2. To migrate custom settings from a previous release of AutoCAD to AutoCAD 2008, choose Start⇨All Programs⇨Autodesk⇨AutoCAD 2008⇨\_\_\_\_\_.

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### Exercise 3-3: Creating a User Profile

For this exercise, open a drawing. Then practice creating a new user profile based on the current one.

1. Choose Tools⇨Options.
2. In the Options dialog box, click the Profiles tab.
3. On the Profiles tab, click the Add to List button.



The new profile that you create is based on the current profile that is active in AutoCAD at the time Add to List is clicked. So if you want the settings from one of your other profiles to be used as the base of the new profile, set the other profile current first before creating the new profile.

4. In the Add Profile dialog box, enter **My Profile for 3D Modeling** in the Profile Name text box.
5. In the Description text box, enter **This is my 3D modeling profile (Last Updated: 11/16/06)**.
6. Click Apply & Close.
7. Click OK to close the Options dialog box.

---

**Exercise 3-4: Setting a User Profile Current and Making Changes**

For this exercise, open the drawing file `Exercise3-4.dwg` from the Chapter 3 folder in the My Practice Drawings folder under My Documents. Practice setting a user profile current and changing a setting in the current user profile.

1. Look at the lights in the drawing; they're most likely displayed in white, black, or some other color that was set up in AutoCAD. Take note of this color for reference later in this exercise.
2. Choose Tools→Options.
3. In the Options dialog box, click the Profiles tab.
4. On the Profiles tab, select the My Profile for 3D Modeling profile from the Available Profiles list.  
  
Before setting the new profile current, take note of the current profile; by default, the profile is <<Unnamed Profile>>.
5. Click Set Current.
6. Click the Drafting tab and then click Lights Glyph Settings.
7. In the Light Glyph Appearance dialog box, click Edit Glyph Colors.
8. In the Drawing Window Colors dialog box, select Yellow from the Color drop-down list.
9. Click Apply & Close to return to the Light Glyph Appearance dialog box.
10. Click OK to return to the Options dialog box.
11. Click OK to close the Options dialog box.  
  
The lights in the drawing are now yellow.
12. Open the Options dialog box again.
13. Set the profile that was current prior to setting the My Profile for 3D Modeling profile current. Click OK to save the changes.
14. Look at the lights in the drawing; the lights are most likely white or black again, or the color that they were before.
15. Close the Options dialog box if it's still open.

---

**Exercise 3-5: Exporting and Importing a User Profile**

Practice exporting and import a profile with a different name.

1. Choose Tools→Options.
2. In the Options dialog box, click the Profiles tab.
3. On the Profiles tab, select the My Profile for 3D Modeling profile from the Available Profiles list.
4. Click Export.
5. In the Export Profile dialog box, browse to the Chapter 3 folder in the My Practice Drawings folder under My Documents.
6. Enter the name **My Profile** in the File Name text box and click Save.
7. In the Options dialog box on the Profiles tab, click Import.

8. In the Import Profile dialog box, browse to the Chapter 3 folder in the My Practice Drawings folder under My Documents.
9. Select the file `My Profile.arg` and click Open.
10. In the Import Profile dialog box, change the name in the Profile Name text box to **My Profile - Imported**.
11. Enter **My Profile Imported is a copy of My Profile** in the Description text box.
12. Select the Include Path Information option to include the paths that the profile was exported with.
13. Click Apply & Close.
14. Set the My Profile - Imported profile current.  
This profile is no different from the My Profile profile.
15. Close the Options dialog box if it's still open.

## Using Workspaces

Workspaces are a relatively new concept in AutoCAD and were once part of the user profile. Workspaces control the display of user interface elements like toolbars, menus, and dockable windows (palettes) in AutoCAD; they control where a user interface element is positioned on screen and which ones are shown or hidden. Workspaces are stored in the current customization file of AutoCAD, which is named `acad.cui` and is located in the folder `C:\Documents and Settings\user name\Application Data\Autodesk\AutoCAD 2008\R17.1\enu\Support`.

AutoCAD comes with three different workspaces that are each oriented towards different tasks. The following table shows which workspaces come with AutoCAD, along with a brief description of what tasks each workspace is targeted at.

<i>Workspace Name</i>	<i>Description</i>
3D Modeling	Displays the most commonly used interface elements that are associated with creating and modifying 3D objects.
AutoCAD Classic	Displays the most commonly used interface elements that are associated with 2D drafting and other general tasks in AutoCAD. Mainly it's all the same user interface elements that were displayed by default with AutoCAD 2005 and earlier.
2D Drafting & Annotation	Displays a subset of the available user interface elements that AutoCAD offers that are used for 2D drafting and working with annotation objects. The difference between this workspace and the AutoCAD Classic is that this one displays the Dashboard and many of the 2D-drafting-oriented panels, instead of toolbars like the AutoCAD Classic workspace does.

After you have created your own workspaces or you want to use one that comes with AutoCAD, you must set it current in order to restore the settings that are saved with the workspace. Here's how you can choose a workspace

- ✓ Choose Tools↔Workspaces and select a workspace to set it current.
- ✓ Select a workspace from the Workspaces drop-down list on the Workspaces toolbar.

- ✓ Enter **workspace** at the command prompt, use the option `setCurrent`, and enter the name of a workspace to set current.
- ✓ Use the Customize User Interface editor to set a workspace current.

To save your workspace, choose `Tools`→`Workspaces`→`Save Current` and enter a name for the workspace.



3. Instead of choosing `Tools`→`Workspaces`→`Save Current`, how else can you save a workspace?
  - A. Enter **createws** at the command prompt and enter a name for the workspace.
  - B. Select `Save Current As` from the `Workspaces` drop-down list on the `Workspaces` toolbar and enter a name for the workspace.
  - C. Enter **workspace** at the command prompt, choose the `SAveas` option, and enter a name for the workspace.
  - D. Select `Workspace Settings` from the `Workspaces` drop-down list on the `Workspaces` toolbar.



### Exercise 3-6: Creating a New Workspace and Setting a Workspace Current

Practice displaying a few user interface elements and creating a new workspace by using the `Tools` menu. After the workspace is created, you alter the user interface and then restore the elements that you alter by using the workspace that you create.

1. Choose `Tools`→`Palettes`→`QuickCalc`.  
The `QuickCalc` palette is displayed.
2. Right-click one of the displayed toolbars and choose `Modeling`.  
The `Modeling` toolbar is displayed.
3. Choose `Tools`→`Workspaces`→`Save Current As`.
4. In the `Save Workspace` dialog box, enter **My DesignSpace** in the `Name` text box.
5. Click `Save`.  
The workspace is created.
6. Close the `QuickCalc` palette and the `Modeling` toolbar by using the same process you used to open them.
7. Choose `Tools`→`Workspaces`→`My DesignSpace`.  
The `QuickCalc` palette and `Modeling` toolbar redisplay where they were before you closed them.

## Workspace settings

You can organize workspaces on the `Tools` menu or the `Workspaces` drop-down list on the `Workspaces` toolbar. To do so, you use the `Workspace Settings` dialog box (see Figure 3-6). The `Workspace Settings` dialog box allows you to insert and remove separators that organize groups of workspaces. Along with organizing workspaces, you can control which workspaces you want to display and in which order they are displayed.

**Figure 3-6:**  
The  
Workspace  
Settings  
dialog box  
allows you  
to organ-  
ize your  
workspaces.



### Exercise 3-7: Organizing Your Workspaces



Practice adding separators to the workspaces list to organize workspaces in groups, reposition workspaces, and control which workspaces are displayed on the Tools menu or in the Workspaces drop-down list on the Workspaces toolbar. You can also enable the automatic saving option to force the saving of a workspace when switching between different workspaces and set the default workspace for the My Workspace feature. Before you start this exercise, be sure that you have completed Exercise 3-6.

1. Choose Tools⇨Workspaces⇨Workspace Settings.
2. In the Workspace Settings dialog box, select the My DesignSpace workspace from the Menu Display and Order list.
3. Click the Move Up button until the My DesignSpace is listed at the top of the list.
4. Select the workspace right below My DesignSpace and click the Add Separator button to insert a separator.
5. Select the check boxes next to the other workspaces except for 3D Modeling, My DesignSpace, and the separator that you just inserted.
6. Under the When Switching Workspaces section, select the Automatically Save Workspace Changes option.

Now if you make a change to the current layout of the user interface, the changes are saved automatically to the current workspace before switching to the requested workspace.

7. Click the My Workspace drop-down list and select My DesignSpace.
8. Click OK to save the changes.
9. Choose Tools⇨Workspaces.

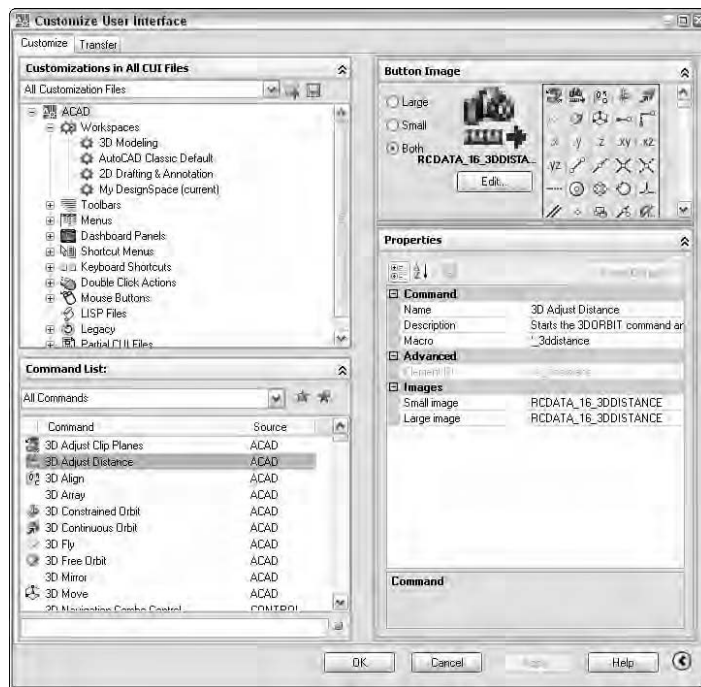
Only two workspaces are being displayed and are separated by a separator bar. The My DesignSpace workspace is also listed at the top of the Workspace submenu.

10. On the Workspaces toolbar, click the My Workspace button to set the My DesignSpace workspace current.

## Customizing AutoCAD and Its User Interface

In AutoCAD, you can customize the user interface by creating new commands and placing them on toolbars, which gives you quick access to repetitive tasks. One of the most popular ways of customizing AutoCAD is to create a custom toolbar that contains many of the standard buttons from other toolbars that you use. To create a custom toolbar or to create a new command to place on an existing toolbar, you use the Customize User Interface editor (see Figure 3-7). This editor not only allows you to create toolbars and custom commands, but it lets you modify the actions that happen when you double-click objects in the drawing window, the context-sensitive menus that display when you right-click in the drawing window, and many other user interface elements.

**Figure 3-7:**  
The  
Customize  
User  
Interface  
editor  
allows you  
to, well,  
customize  
the user  
interface.



### Customizing the user interface

The Customize User Interface editor allows you to take control of the user interface and really customize it to the way you want to work. The user interface elements that are displayed in AutoCAD (such as toolbars, menus, and even the behavior of things like shortcut key combinations and double-click actions) are defined as part of a customization (CUI) file. AutoCAD allows you to load multiple different customization files at a time to load the tools that you need to get your work done as efficiently as possible.

The Customize User Interface editor is used for two different tasks: to customize the user interface and to transfer customization to and from a customization file. In most cases, you work on the Customize tab, unless you're migrating from an older release of AutoCAD, in which case you might find yourself using the Transfer tab. The Customize tab includes an area to add new user interface elements and other areas to create custom commands and modify the properties of a command or user interface element.

Although this section covers only how to create a basic custom command and create a new toolbar, you can find more about customizing AutoCAD in the Customization Guide of the Online Help system. To access the information on customizing the interface,

choose Tools→Customize→Interface and click the Help button when the Customize User Interface editor is displayed.



### Exercise 3-8: Creating a Custom Command

Practice creating a custom command that draws a red 5 x 5 rectangle.

1. Choose Tools→Customize→Interface.
2. Click the Customize tab and click Create a New Command under the Command List pane.
3. Click the Name field under the Properties pane and enter **My Rectangle**.
4. In the Description field, enter **Creates a Red 5x5 Rectangle**.
5. In the Macro field, enter the following:
 

```
^C^C._rectang;\@5,5;._change;_l; ;_p;_c;1; ;
```
6. Click OK to save the command to the CUI file and close the Customize User Interface editor.



### Exercise 3-9: Creating a Custom Toolbar

Practice creating a custom toolbar and add a custom and standard command to it.

1. Choose Tools→Customize→Interface.
2. Click the Customize tab and right-click Toolbars in the Customizations In pane.
3. Click New Toolbar.
4. Click the Name field under the Properties pane and enter **My Tools**.
5. In the Description field, enter **My Custom Tools**.
6. In the Commands List pane, select Custom Commands from the Categories drop-down list.
7. Select the My Rectangle command and drag it from the command list up to the My Tools node under Toolbars in the Customizations In pane. Release the mouse button when the cursor is over the node for the My Tools toolbar.
8. Scroll down in the Command List pane until you find Rectangle.
9. Add the Rectangle command to the My Tools toolbar.
10. Click OK to save the changes to the toolbar and the CUI file and then close the Customize User Interface editor.

The new toolbar My Tools floats on-screen near the upper-left corner of AutoCAD. Click the toolbar buttons to test them and pick as many points in the drawing that you're requested to select. The My Rectangle command prompts you for one point, but the Rectangle command prompts you for two points.

## Other methods of customizing AutoCAD

AutoCAD is one of the most widely used Computer-Aided Drafting (CAD) applications for a couple different reasons. First, it was one of the most affordable CAD drafting applications on a personal computer back in the early 1980s, when most drafting applications ran on mainframes that took up a lot of space. Second, AutoCAD was back then and still is today the world's most customizable drafting application on the

market. AutoCAD not only allows you to change the way tools are arranged on-screen, but it allows you to actually create custom programs that can alter the workflow of your projects. So you can adapt AutoCAD to your business instead of adapting your business to work with AutoCAD.

By customizing AutoCAD, you can create your own commands that might insert or populate information in your title blocks or change the appearance of lines in the drawing through a custom linetype. The following list mentions many of the features you can customize in AutoCAD:

- ✓ Drawing templates
- ✓ Blocks
- ✓ Tool palettes
- ✓ Custom menus
- ✓ Custom shapes
- ✓ Custom linetypes
- ✓ Custom hatch patterns
- ✓ Command aliases
- ✓ AutoLISP and Visual LISP
- ✓ Visual Basic and Visual Basic for Applications
- ✓ VB .NET or C#
- ✓ ObjectARX

To find out more about customizing AutoCAD, check out the Customization Guide in the Online Help system by choosing Help⇨Help and then clicking the Customization Guide link. In addition to the Customization Guide, AutoCAD offers an Online Help file called Developer Help, which covers many of the different programming options that are available. Open the Developer Help file by choosing Help⇨Additional Help⇨Developer Help.

## *Answers to Practice Opportunities*

### **1. What is the Options dialog box used for?**

You use the Options dialog box to define some of the general settings that are used globally in AutoCAD, define how files are saved or plotted, and define settings that control the behavior of selecting objects.

### **2. To migrate custom settings from a previous release of AutoCAD to AutoCAD 2008, choose Start⇨All Programs⇨Autodesk⇨AutoCAD 2008⇨Migrate Custom Settings.**

### **3. Instead of choosing Tools⇨Workspace⇨Save Current, how else can you save a workspace?**

B, C. Do one of the following and then enter a name for your workspace:

- Select the Save Current As option from the Workspaces drop-down list on the Workspaces toolbar.
- Enter **workspace** at the command prompt.



# Part II

## Going from 2D to 3D

The 5<sup>th</sup> Wave

By Rich Tennant



"That reminds me - I have to figure out exploding the block with AutoCAD."

## *In this part . . .*

**A**lthough some of the concepts that you already know about 2D drafting have equivalent concepts when talking about 3D modeling, a number of concepts are unique to working in 3D. Before you start modeling, you should know the differences between the three types of 3D models that AutoCAD allows you to create: wireframe, surface, and solid. As you start making the transition from two dimensions to three dimensions, you find out how to specify coordinate values using a Z-direction and how to manipulate the user coordinate system to draw objects on different planes.

2D drawings are primarily viewed using just pan and zoom because they're flat, but with 3D models, you have to navigate around the model from all different angles and directions. You use visual styles when viewing a 3D model to control how the edges and faces of the objects that make up your model are displayed. After getting a good foundation of inputting coordinates and navigating a 3D model, you discover how to transition what you already know about creating and modifying 2D objects to create and modify 3D objects using 2D objects.

## Chapter 4

# Transitioning from 2D to 3D

### *In This Chapter*

- ▶ Understanding the three different types of 3D models
- ▶ Specifying coordinates in the three directions (X,Y,Z)
- ▶ Working with the user coordinate system (UCS)

**M**odeling in AutoCAD poses some interesting challenges if you're completely new to 3D. The first thing that you need to understand is the different types of 3D models that AutoCAD allows you to create. The second thing to know is which type of modeling suits your needs. After understand the different modeling types available for creation, it's time to introduce working with the Z plane because 3D modeling introduces an additional axis when entering coordinates that you most likely haven't dealt with if you've created only 2D drawings in AutoCAD. When working with the Z axis, you have additional options for entering coordinate values when creating and modifying objects. The exercises in this chapter give you an understanding of how to enter coordinate values and work with the user coordinate system (UCS) in AutoCAD when working with 3D models.

## *3D Modeling Types Available in AutoCAD*

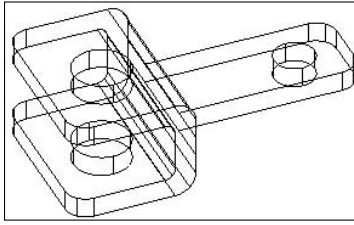
AutoCAD allows you to create three different types of 3D models: wireframe, surface, and solid. Each model type has strengths and weaknesses. When you want to create a presentation drawing of some sort, surface and solid 3D models are what you need; to create a solid or surface model, you can use a wireframe model as a template.

### *Wireframe model*

Wireframe models are often referred to as 2½D drawings because they aren't 2D drawings in the true sense; they're created with 2D objects but in 3D space. Wireframe models aren't true 3D models because they lack faces, which allow objects that are beyond to be hidden when a 3D visual style is applied or a rendering is done. You can think of a wireframe model as being like a wire coat hanger or a chain link fence. Wireframe models are made up of standard 2D objects that are given an additional coordinate value for the Z direction.

You can use most 2D objects when creating a wireframe model, but objects such as lines, arcs, and circles are commonly used. The 2D objects are drawn in 3D using full X, Y, and Z coordinate values rather than just X and Y coordinate values. Figure 4-1 shows a wireframe model of a bracket that connects two parts together. The wireframe model is shown with the 3D Hidden visual style; all the individual 2D objects used to create the model are shown.

**Figure 4-1:**  
A wireframe  
model of a  
bracket.

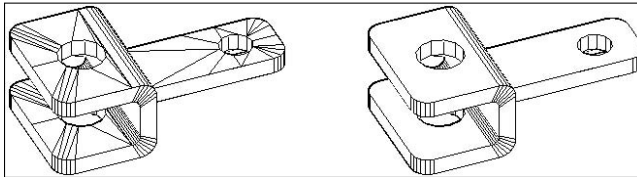


## Surface model

Surface models are created from 3D objects called *faces*, which are true 3D objects in the sense that they contain a surface that can hide objects beyond them when a 3D visual style is applied or a rendering is done. Faces are created with three or four sides. Although the surface of a face can hide objects that are beyond it in the current view, they require some cleanup when shared edges are involved to provide a clean look that makes individual surfaces seem like a much larger surface.

AutoCAD offers a number of commands that allow you to create individual faces or primitive objects such as boxes, spheres, and cones. Figure 4-2 shows two surface models of the same bracket; the left model doesn't have any of its edges cleaned up, whereas the right model has been cleaned up. The surface model is shown with the 3D Hidden visual style, and the objects closest to you in the current view hide the objects beyond them (unlike the wireframe model). Part IV covers surface models in greater depth.

**Figure 4-2:**  
Two surface  
models of a  
bracket. The  
model on  
the right is  
cleaned up.

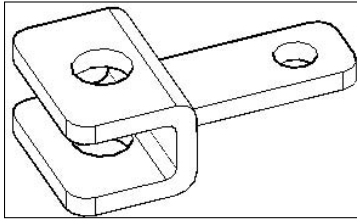


## Solid model

Solid models are created from 3D objects called *solids*, which are true 3D objects similar to faces that are used to make up surface models. 3D solids in AutoCAD are very different from surface models; these objects aren't flat surfaces. Also unlike surface models, solid models don't require additional cleanup of shared edges of the model because a 3D solid object is one continuous object instead of being made up of many different faces.

AutoCAD offers a number of commands that allow you to create primitive objects such as boxes, spheres, and cones along with editing the faces, edges, and bodies of a solid in many different ways. Figure 4-3 shows a solid model of a bracket. The solid model is shown with the 3D Hidden visual style, and the objects closest to you in the current view hide the objects beyond them (like a surface model does). Part III of this book covers solid models in greater depth.

**Figure 4-3:**  
A solid  
model of a  
bracket.



## Going Above the Plane

When creating 2D drawings, you create all objects on the working plane by using absolute (X,Y), relative (@X,Y), and polar (@distance<angle) coordinate values. As you create 3D models, you need to also supply an additional coordinate value along the Z axis (which in most cases gives an object height). Like with 2D drafting, you can input coordinate values by manually entering them at the command prompt, picking a point using object snaps, or using polar or object snap tracking. As you create new objects in a drawing, you can preset the height of the working plane so you don't have to specify a Z coordinate value when the coordinates are on a specific plane.



1. How can you specify a coordinate value?
  - A. Pick a point in the current viewport.
  - B. Enter a coordinate value at the command prompt.
  - C. Use Snap mode to pick a point on an object.
  - D. Use direct distancing and enter a distance value to specify the next point.

## Entering coordinate values at the command prompt

Understanding how to enter coordinate values when working in 3D can save you time and help you produce accurate models. When creating 2D objects, you mainly use three different coordinate values: absolute, relative, and polar. Polar coordinate entry is different when working with 3D models because you have two different variations to work with: spherical and cylindrical.

### ***Absolute coordinates (X,Y,Z)***

Absolute coordinates in 3D are the same as absolute coordinates in 2D with the exception of adding a Z coordinate value. Absolute coordinate values are expressed in the format X,Y,Z and are based on the current origin of the drawing. The following is an example of drawing a line in 3D space using absolute coordinate entry:

```
Command: line  
Specify first point: 0,0,0  
Specify next point or [Undo]: 2.5,4,7.865
```



When prompted to specify a point, you can enter @ at the command prompt to use the previous point specified in a drawing.

### Relative coordinates (@X,Y,Z)

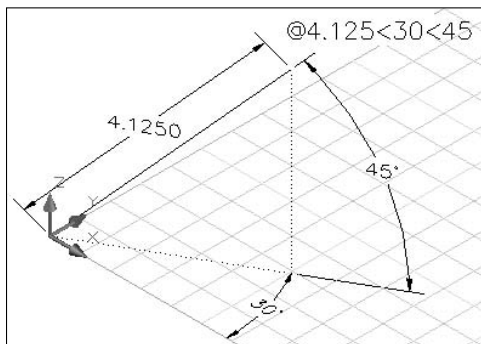
Relative coordinates in 3D are the same as relative coordinates in 2D with the exception of adding a Z coordinate value. Relative coordinate values are expressed in the format @X,Y,Z and are based on the most recently entered coordinate value specified in a drawing. The following is an example of drawing a line in 3D space using relative coordinate entry:

```
Command: line
Specify first point: 1.25,5.675,2.15
Specify next point or [Undo]: @5,5,0.125
```

### Spherical coordinates (@distance<angle<anglefromXY)

The spherical coordinate format is the first of two possible coordinate entry formats based on relative polar coordinate entry. Spherical coordinate values are expressed in the format @distance<angle<anglefromXY. The following is an example of drawing a line in 3D space using spherical coordinate entry. The results of entering a spherical coordinate value to create a line are shown in Figure 4-4.

```
Command: line
Specify first point: 0,0,0
Specify next point or [Undo]: @4.125<30<45
```



**Figure 4-4:** A spherical coordinate entry used to create a line in 3D space.

### Cylindrical coordinates (@XYdistance<angle,Z)

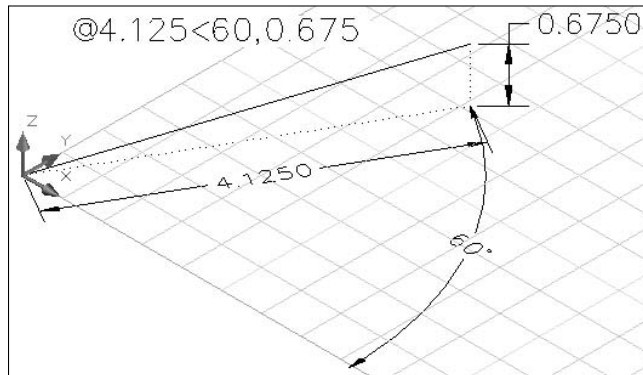
The cylindrical coordinate format is the second of two possible coordinate entry formats based on relative polar coordinate entry. Cylindrical coordinate values are expressed in the format @XYdistance<angle,Z. The following is an example of drawing a line in 3D space using cylindrical coordinate entry. Figure 4-5 shows a line that was created by entering a cylindrical coordinate value.

```
Command: line
Specify first point: 0,0,0
Specify next point or [Undo]: @4.125<60,0.675
```



When using Dynamic Input to enter coordinate values, avoid using the cylindrical coordinate entry method because it doesn't work very well. To disable Dynamic Input, click the DYN button on the status bar or press F12.

**Figure 4-5:**  
A cylindrical  
coordinate  
entry used  
to create a  
line in 3D  
space.



### Exercise 4-1: Creating a Wireframe Model Using Coordinate Entry

For this exercise, open the drawing `Exercise4-1.dwg` from the Chapter 4 folder in the My Practice Drawings folder under My Documents. Practice using absolute, relative, and cylindrical coordinate entry methods to draw a wireframe of a wedge-shaped object.

1. On the status bar, right-click the DYN button and choose Off.
2. Choose Draw→Line to start the LINE command.
3. At the Specify first point: prompt, enter `0,0` and press Enter.
4. At the Specify next point or [Undo]: prompt, enter `5,0` and press Enter.
5. At the Specify next point or [Undo]: prompt, enter `@2<90` and press Enter.
6. At the Specify next point or [Close/Undo]: prompt, enter `@5<180` and press Enter.
7. At the Specify next point or [Close/Undo]: prompt, enter `c` and press Enter to close the object.
8. Start the LINE command again and start the line at 0,0.
 

You can quickly repeat the last command by pressing Enter or the spacebar when no command is currently active.
9. At the Specify next point or [Undo]: prompt, enter `@5<0,0.675` and press Enter.
10. At the Specify next point or [Undo]: prompt, enter `@2<90` and press Enter.
11. At the Specify next point or [Close/Undo]: prompt, enter `0,2` and press Enter. Press Enter again to end the command.
12. Start the LINE command again and start the line at 5,0.
13. At the Specify next point or [Undo]: prompt, enter `@0,0,0.675` and press Enter. Press Enter again to end the command.
14. Start the LINE command one final time and start the line at 5,2.
15. At the Specify next point or [Undo]: prompt, enter `5,2,0.675` and press Enter. Press Enter again to end the command.
16. Save the drawing with the name `Completed4-1.dwg` to the Chapter 4 folder in the My Practice Drawings folder under My Documents and close the drawing.



## Object snaps

*Object snaps* increase the accuracy of creating and modifying objects in 2D and 3D drawings. For the most part, object snaps are the same in 3D as they are in 2D; you can draw to the midpoint or from the endpoint of an object or use any of the other object snaps to snap to points on objects. 3D object snaps differ from 2D in two ways:

- ✓ The Apparent Intersection object snap snaps to the point where two objects intersect each other in the current view, but in all actuality they never intersect each other.
- ✓ You can force objects to the current elevation even if you're picking a snap on an object with a different Z coordinate value.



2. How do you set running object snaps?
- A. Hold down Shift, right-click, and choose the object snaps.
  - B. Right-click the OSNAP button on the status bar, choose Settings, and then set the object snaps you want to use.
  - C. Choose Tools⇨Drafting Settings, click the Object Snaps tab, and then set the object snaps you want to use.
  - D. At the command prompt, enter **dsettings** and press Enter. In the Drafting Settings dialog box, click the Object Snaps tab and then set the object snaps you want to use.

The current running object snaps are set on the Object Snap tab in the Drafting Settings dialog box, and the option to replace the Z coordinate value with the current elevation is located on the Drafting tab of the Options dialog box. (Changing the current elevation is covered in the section “Elevating [not levitating] objects,” later in this chapter.) Along with the Replace Z Value with Current Elevation option in the Options dialog box, you can change the appearance of the AutoSnap markers that appear to indicate which type of object snap will be used at the snap point when picked.

---

### Exercise 4-2: Using Object Snaps in 3D Space



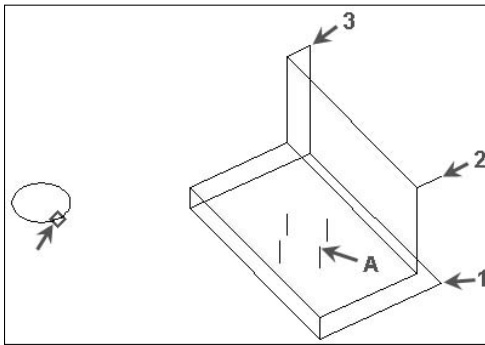
For this exercise, open the drawing `Exercise4-2.dwg` from the Chapter 4 folder in the My Practice Drawings folder under My Documents. Practice using absolute, relative, and cylindrical coordinate entry methods to draw a wireframe of a wedge-shaped object.

1. On the status bar, right-click the OSNAP button and choose Settings.
2. In the Drafting Settings dialog box, click Clear All and then select the Quadrant and Endpoint check boxes.
3. If the Object Snap On check box isn't selected, select it to enable running object snaps.
4. Click the Options button.
5. In the Options dialog box on the Drafting tab, deselect Replace Z Value with Current Elevation.
6. Click OK to return to the Drafting Settings dialog box.
7. Click OK to return to the drawing window.



8. Choose Draw↔Line and pick the points indicated in Figure 4-6. Pick them in the proper order and end the LINE command.

**Figure 4-6:**  
Using object snaps to finish off a wireframe model of an angle bracket used to mount a steel rail to another part.



9. Choose Tools↔Options.
10. In the Options dialog box on the Drafting tab, select Replace Z Value with Current Elevation.
11. Click OK to return to the drawing window.
12. Choose Modify↔Copy.
13. Select the circle that is to the left of the wireframe model and press Enter.
14. At the Specify base point or [Displacement/mOde] <Displacement>: prompt, pick at the quadrant indicated on the circle by the arrow in Figure 4-6.
15. At the Specify second point or <use first point as displacement>: prompt, pick the end of the line indicated by the letter A in Figure 4-6 using the endpoint object snap. Press Enter to end the command.

The circle is copied to the bottom of the line instead of at the end that you selected with the endpoint object snap, and this is because the current elevation of the drawing is 0 (zero).

16. Choose Tools↔Options and click the Drafting tab.
17. On the Drafting tab, deselect Replace Z Value with Current Elevation.
18. Choose Modify↔Move.
19. If the Selecting Subobjects on Solids dialog box is displayed, click Close.
20. Select the Circle that is to the left of the wireframe model and press Enter.
21. At the Specify base point or [Displacement] <Displacement>: prompt, pick at the quadrant indicated on the circle by the arrow in Figure 4-6 using the endpoint object snap.
22. At the Specify second point or <use first point as displacement>: prompt, pick the end of the line indicated by the letter A in Figure 4-6. Press Enter to end the command.

The circle this time moves to the object snap point by letter A because the current elevation is ignored for the point you picked.

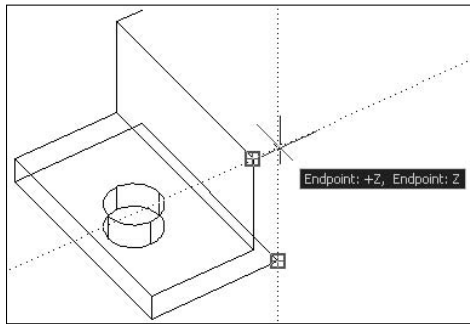
23. Save the drawing with the name Completed4-2.dwg to the Chapter 4 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Object snap tracking

Object snap tracking creates new objects that start at the intersection of two existing points in the drawing or a point along a projected path from an acquired snap point. You can use object snap tracking to track points that run along the XY plane or in the Z direction. If you use object snap tracking in the Z direction, you're provided real-time feedback of the current position along the projected path in a tooltip.

The tooltip indicates whether you're moving in a positive or negative direction from the acquired snap point. To enable object snap tracking, right-click the OTRACK button on the status bar and choose On or press the F11 key. Then, to acquire a point, hover the cursor over a snap point for a moment. After you acquire a snap point, a temporary marker is added to the drawing, and a vector is drawn orthogonally from the marker or along a polar angle. To remove the marker for an acquired point, hover the cursor over the snap point again, and the marker is removed. Figure 4-7 shows tracking a point along a projected path in the Z direction with the tooltip displaying the current Z value in the positive direction.

**Figure 4-7:**  
Tracking a  
point off the  
model in the  
Z direction.



### Exercise 4-3: Using Object Snap Tracking

For this exercise, open the drawing `Exercise4-3.dwg` from the Chapter 4 folder in the My Practice Drawings folder under My Documents. Practice using object snap tracking and object snaps to finish the back of the bracket.

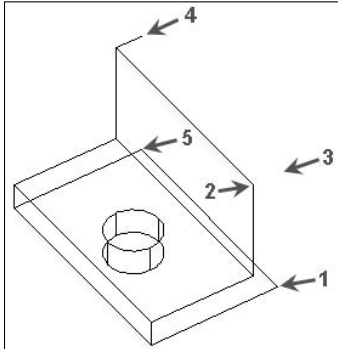


1. On the status bar, right-click the OSNAP button and choose Settings.
2. In the Drafting Settings dialog box, click Clear All and then select the Endpoint and Intersection check boxes.
3. If the Object Snap On check box isn't selected, select it to enable running object snaps.
4. If the Object Snap Tracking On check box isn't selected, select it to enable object snap tracking.
5. Click OK to return to the drawing window.



6. On the status bar, right-click the ORTHO button and choose On.  
You can also press the F8 key to toggle orthomode on and off.
7. Choose Draw↔Line.
8. At the `Specify first point:` prompt, select the point indicated by the number 1 in Figure 4-8.

**Figure 4-8:**  
Finishing off  
a wireframe  
model of  
an angle  
bracket  
used to  
mount a  
steel rail to  
another  
part using  
object snap  
tracking.



9. At the *Specify next point or [Undo]:* prompt, hover the cursor over the point indicated by the number 2 in Figure 4-8 but don't click.  
A marker is placed at the point, indicating that the point has been acquired for object snap tracking.
10. Move the crosshairs to the point indicated by the number 3 in Figure 4-8. The two vectors are projected where the starting point of the line and the acquired point would intersect. Click in the drawing when the object snap tracking tooltip displays the text *Ortho: +Z, Endpoint: Z*.
11. At the *Specify next point or [Undo]:* prompt, pick the endpoint indicated by the number 4 in Figure 4-8.
12. At the *Specify next point or [Close/Undo]:* prompt, pick the endpoint indicated by the number 5 in Figure 4-8. Press *Enter* to end the command.
13. Start the *LINE* command again and draw a line between the corners indicated by the points numbered 2 and 3.
14. Save the drawing with the name *Completed4-3.dwg* to the Chapter 4 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Point filters

Point filters construct coordinate values based on existing points of 2D or 3D objects. Point filters for the most part are obsolete with everything that can be done with object snap tracking, but they can come in handy from time to time when you want to specify a coordinate value that isn't at the intersection of two snap points. Six point filters are available to you when specifying points, but three are for obtaining 2D points (.X, .Y, and .Z) and three are for obtaining 3D points (.XY, .XZ, and .YZ). You can enter point filters at the command prompt, where you're prompted to specify a point, or you can select them from the Object Snap shortcut menu or the Snap Overrides submenu. The following is an example of specifying a coordinate value with a point filter:

```
Command: line
Specify first point: specify a point or enter a coordinate value
Specify next point or [Undo]: enter .XY and press ENTER
of specify a point or enter a coordinate value
(need Z): specify a point or enter a Z coordinate value
```

## *Elevating (not levitating) objects*

When working in 3D, the working plane by default is at the elevation of 0 (zero). So when a specific Z coordinate value isn't entered, the elevation value is substituted when you manually enter a coordinate value or pick a point in the drawing without using object snaps. When the object snap setting Replace Z Value with Current Elevation (located on the Drafting tab of the Options dialog box) is enabled, points selected using object snaps are forced to substitute the current elevation value for the selected Z coordinate value; otherwise, if the option is disabled, no substitution occurs when picking a snap point. To set the current elevation, you use the ELEV command.

---

### Exercise 4-4: Changing the Current Elevation of a Drawing

For this exercise, open the drawing `Exercise4-4.dwg` from the Chapter 4 folder in the My Practice Drawings folder under My Documents. Practice changing the current elevation in the drawing and create a wireframe model of a cube.

1. Choose Draw↔Line.
2. At the Specify first point: prompt, enter **0,0** and press Enter.
3. At the Specify next point or [Undo]: prompt, enter **500,0** and press Enter.
4. At the Specify next point or [Undo]: prompt, enter **500,500** and press Enter.
5. At the Specify next point or [Close/Undo]: prompt, enter **0,500** and press Enter.
6. At the Specify next point or [Close/Undo]: prompt, enter **c** and press Enter to close the object.
7. At the command prompt, enter **elev** and press Enter.
8. At the Specify new default elevation <0.0000>: prompt, enter **500** and press Enter twice.
9. Repeat Steps 2 through 6 with the new elevation.  
The lines being created are 500 units above the original lines that you drew in Steps 2 through 6.
10. Use the LINE command and draw vertical lines in the Z direction that connect the bottom of the cube to the top of the cube.
11. Save the drawing with the name `Completed4-4.dwg` to the Chapter 4 folder in the My Practice Drawings folder under My Documents and close the drawing.



If you need to work with many different elevations in a drawing, you might want to create a named user coordinate system (UCS) so you can quickly switch between different elevations. The next section describes how to create a custom UCS.

## *Working with the Coordinate Systems*

AutoCAD uses a coordinate system to determine where objects are created in a drawing, but many users don't know how to harness the full power of it. As you create 2D drawings, for the most part the coordinate system is left untouched except maybe in some cases where you might move the origin in a very large drawing to make entering

coordinate values easier. However, when working in 3D space the coordinate system plays a much bigger part in how you create and modify objects.

You can manipulate the coordinate system in order to create objects going in different directions or even to continue drawing objects on the face of an existing 3D object. As an example, you can align the coordinate system with the face of a 3D object to create a hole in the surface. With that technique, you don't have to draw the object first on the XY plane and then modify the object by rotating/moving/aligning it into its final position.

## *Understanding the coordinate systems*

AutoCAD utilizes two different coordinate systems when creating and modify objects in a drawing. These two coordinate systems are known as *world* and *user*.

### *World coordinate system (WCS)*

The world coordinate system (WCS) is controlled by AutoCAD and is fixed, so you can't alter it like the user coordinate system. The WCS stores and defines all objects that you create in a drawing. When working in a 2D drawing, you commonly work with the WCS. The X axis of the WCS runs horizontally (left and right) across the drawing, and the Y axis runs vertically (up and down) across the drawing. The positive X direction is defined to go to the right, and the positive Y direction goes up.

### *User coordinate system (UCS)*

The user coordinate system (UCS) is controlled by you as the user. You can manipulate the coordinate system, which allows you to easily create and modify objects. The current UCS affects how coordinate values are entered, the orientation that objects are created in, and how many of the drawing aids such as object snap tracking and the grid work. You can change the UCS manually and dynamically as needed. When you manually define the UCS, you can create it as a named UCS so you can use it again.



3. When do you use the WCS, and when do you use the UCS?

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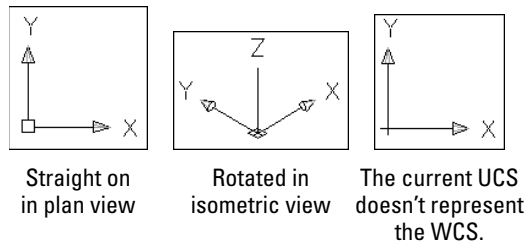
## *Getting your bearings from the UCS icon*

The UCS icon to a drafter is like a compass to a sailor; never start drawing without knowing which direction you're heading in. As you work in a drawing, the UCS icon is there to show you which direction the positive X, Y, and Z axes are going in and gives you a heads-up about whether you're on the Model tab or a paper space layout tab. As you change tabs or view the drawing with a 2D or 3D visual style, AutoCAD displays a slightly different-looking UCS icon to let you know that these conditions are current in the drawing. The appearance of the UCS icon can be customized in many different ways, which I cover later in this section.

### *UCS icon in Model space with 2D wireframe visual style*

As you view a drawing in plan view (looking down at the XY plane), the UCS icon indicates which direction the positive X and Y axes are going in. When viewing a drawing in an isometric view, the UCS icon appears rotated, and it shows you which direction the positive X, Y, and Z axes are going in. Figure 4-9 shows the UCS icon in three different situations on the Model tab: straight on in plan view, rotated in an isometric view, and when the current UCS doesn't represent the WCS. If a box appears at the intersection of the two or three axes of the UCS icon, you're looking at the drawing from the WCS, and if no box is displayed, you aren't viewing the drawing from the WCS.

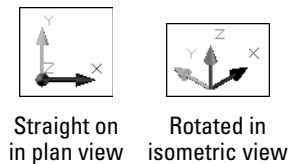
**Figure 4-9:**  
Different views of the UCS icon when the 2D wireframe visual style is in use.



### *UCS icon in Model space with 3D wireframe visual style*

As you view a drawing in plan view (looking down at the XY plane), the UCS icon indicates which direction the positive X and Y axes are going in and the direction of the Z axis, which is coming towards you by default but could be also moving away from you. When viewing a drawing in an isometric view, the UCS icon appears rotated on-screen so you see which direction the positive X, Y, and Z axes are going in. Figure 4-10 shows the UCS icon in two different situations on the Model tab: straight on in plan view and rotated in an isometric view.

**Figure 4-10:**  
Different views of the UCS icon when a 3D visual style is in use.



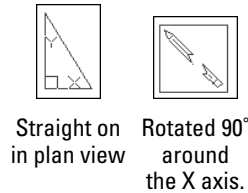
### *UCS icon in paper space layout*

If a paper space layout is active, the UCS icon is shown as a right triangle (with or without a box), and it indicates which direction the positive X and Y axes are going in, or else it looks like a broken pencil. Because a paper space layout represents a virtual sheet of paper, the things that affect the appearance of the UCS icon on the Model tab don't completely apply to the UCS icon on a paper space layout. Visual styles can't be set on a paper space layout, nor can the view be displayed in isometric.

The only thing that affects the UCS icon is rotating the current UCS. Like in Model space with the 2D wireframe visual style, a box on the UCS icon indicates that the UCS icon

represents the WCS. If no box shows on the icon, the WCS isn't currently represented. The UCS icon displayed on a paper space layout can also be displayed as a broken pencil, which indicates that the UCS icon has been rotated so far that it can't be correctly shown. Figure 4-11 shows the UCS icon in two different situations on a paper space layout: straight on in plan view and rotated 90 degrees around the X axis.

**Figure 4-11:**  
Different views of the UCS icon when a paper space layout is active.



Straight on in plan view      Rotated 90° around the X axis.

## Customizing the display of the UCS icon

How you're viewing the current drawing can affect the UCS icon, but you do have the ability to control whether the UCS icon is displayed at all or whether it appears only when you're in a certain context. You can also control whether it's always displayed in the lower-left corner of a drawing window or whether it appears at the origin of a drawing when the origin is in the current viewport.

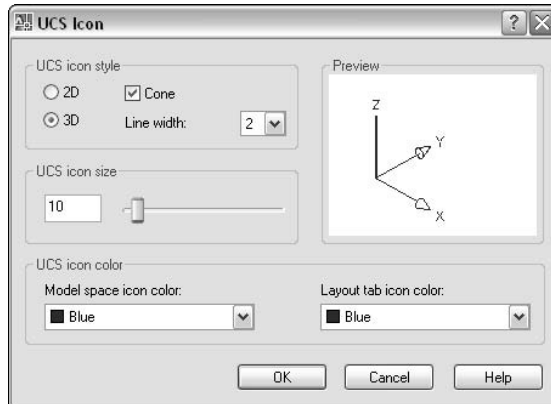
The UCS icon is displayed in the lower-left corner of the drawing window when it can't be displayed at the origin of the drawing. The origin of the drawing is usually 0,0, but it can be changed for the current UCS. Choose View→Display→UCS Icon→On. The context in which the UCS icon is displayed is controlled by the options under the section Display UCS Icon on the 3D Modeling tab of the Options dialog box.



4. Besides choosing View→Display→UCS Icon→On, how can you change where the UCS icon is displayed?
- Right-click the drawing window and choose UCS Icon On from the shortcut menu.
  - Choose or View→Display→UCS Icon→Origin.
  - Enter **uscicon** at the command prompt and designate where the UCS icon is displayed in the drawing window.
  - Choose View→Display→UCS Icon→Enable or View→Display→UCS Icon→@0,0.

Along with specifying when and where the UCS icon is displayed, you can control how it looks when the 2D Wireframe visual style is applied to the current viewport. The UCS Icon dialog box (see Figure 4-12) customizes the appearance of the UCS icon when the 2D Wireframe visual style is applied to the current viewport. You can specify whether the UCS icon has a 2D or 3D look to it, its size relative to the height of the drawing window, and its color when it's on the Model or a layout tab. To display the UCS icon dialog box, choose View→Display→UCS Icon→Properties.

**Figure 4-12:**  
The UCS  
Icon dialog  
box controls  
the display  
of the UCS  
icon when  
the 2D  
wireframe  
visual style  
is applied to  
the current  
viewport.



### Exercise 4-5: Customizing the Display of the UCS Icon

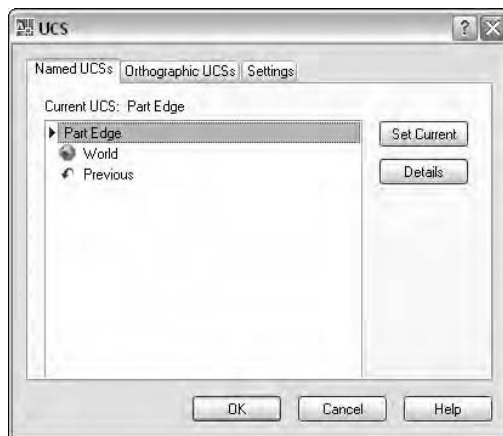
Practice customizing the display of the UCS icon with the UCS Icon and Options dialog boxes.

1. Create a new drawing with the drawing template `Acad.dwt`.
2. If the UCS icon isn't displayed, choose `View` → `Display` → `UCS Icon` → `On`.  
Take note of how the UCS icon looks; by default, it's black.
3. Choose `View` → `Display` → `UCS Icon` → `Origin` to toggle the position of the UCS icon so it's displayed at 0,0 or in the lower-left corner of the current viewport.
4. Choose `View` → `Display` → `UCS Icon` → `Properties`.
5. In the UCS Icon dialog box, select 2 from the Line Width drop-down list.
6. Slide the slider in the UCS Icon Size section until the value 10 is displayed in the text box or enter **10** in the text box.
7. Select Blue from the Model Space Icon Color drop-down list under the UCS Icon Color section.
8. Click OK.
9. Choose `Tools` → `Options`.
10. On the 3D Modeling tab, deselect Display in 3D Perspective Projection.
11. Click OK to exit the Options dialog box.  
The UCS icon is now blue in color and the lines are much heavier, which makes it stand out better.
12. Choose `View` → `Visual Styles` → `3D Hidden`.  
The UCS icon now changes from a flat line drawn object to what looks more like a 3D object with three different colors.
13. Right-click the drawing window and choose Zoom.
14. Right-click the drawing window again and choose Perspective.
15. Press Enter.  
The UCS icon now disappears from the viewport because you deselected the Display in 3D Perspective Projection option in Step 10.
16. Close the drawing and don't save the changes.



## Modifying the UCS

Working on a 2D drawing is much simpler than working on a 3D model because you're working in a top-down view most of the time, so the origin and direction of the UCS are always going in the same direction from drawing to drawing. As you start working with 3D drawings, you will need to draw an object on the face of another object. To move or change the orientation of the UCS in the viewport, you use the UCS command. If you need to refer to the same UCS a number of times, you can save a UCS with a name so you can restore it later when working with the same objects or view. The UCS dialog box (see Figure 4-13) allows you to manage the named UCSs in a drawing.



**Figure 4-13:**  
The UCS dialog box allows you to manage named UCSs.

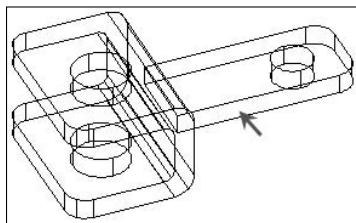
### Exercise 4-6: Changing the UCS and Creating a Named UCS



For this exercise, open the drawing `Exercise4-6.dwg` from the Chapter 4 folder in the My Practice Drawings folder under My Documents. Practice manipulating the UCS by aligning the UCS with the face of an object and the current view. You also create a named UCS and return to the WCS.

1. Choose `Tools` → `New UCS` → `Face`.
2. At the `Select face of solid object:` prompt, select the face indicated by the arrow in Figure 4-14 to move the UCS and change its orientation to match the face.

**Figure 4-14:**  
Moving and orientating the UCS with the selected face of the bracket.

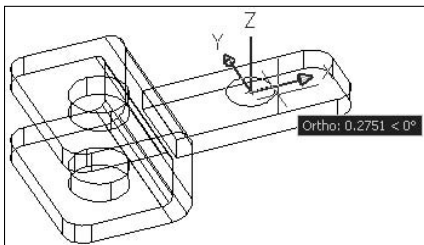


3. At the Enter an option [Next/Xflip/Yflip] <accept>: prompt, press Enter to accept the change to the UCS.
4. Choose Tools→Named UCS.
5. In the UCS dialog box, right-click Unnamed in the list of named UCSs and choose Rename from the shortcut menu.
6. Enter **Part Edge** for the name of the named UCS.
7. Click OK to exit the UCS dialog box.
8. Choose Tools→New UCS→View.  
The UCS remains in its previous location, and its orientation is updated to be parallel to the screen.
9. Choose Tools→Named UCS.
10. Select Part Edge from the named UCSs list and click Set Current.
11. Click OK to exit the UCS dialog box.  
The UCS returns to the orientation saved with the named UCS Part Edge.
12. Choose Tools→New UCS→World.  
The UCS icon returns to the origin and orientation of the WCS in the drawing.
13. Save the drawing with the name `Completed4-6.dwg` to the Chapter 4 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Manipulating the UCS dynamically

Creating a new UCS each time you want to create a new object on an existing 3D object can consume a lot of time switching between different UCSs, and for this reason, AutoCAD comes with a feature called *Dynamic UCS*. Dynamic UCS allows you to create a temporary UCS on the face of an object by simply positioning the crosshairs over the face of the object and clicking. To enable/disable Dynamic UCS, you can click the DUCS button on the status bar or press the F6 key. When Dynamic UCS is enabled and you're requested to specify a point, the face of the object that the crosshairs is over becomes highlighted to indicate that if you click now, the UCS dynamically changes to align with that face (see Figure 4-15).

**Figure 4-15:** Making a UCS dynamically on the fly when creating an object over the face of a 3D solid.



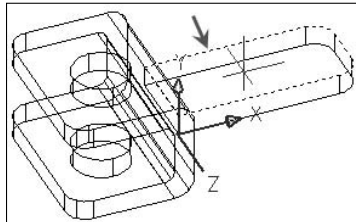


### Exercise 4-7: Using Dynamic UCS

For this exercise, open the drawing `Exercise4-7.dwg` from the Chapter 4 folder in the My Practice Drawings folder under My Documents. Practice creating an object on the face of a 3D solid using Dynamic UCS.

1. On the status bar, right-click the DUCS button and choose On.  
Take note of the current position and orientation of the UCS icon in the corner of the face along the edge of the part.
2. Choose `Draw` → `Circle` → `Center, Radius`.
3. Position the crosshairs over the face indicated by the arrow in Figure 4-16.  
The edges of the face become highlighted.

**Figure 4-16:**  
Position the crosshairs over a face to use for Dynamic UCS.



4. With the edges of the face highlighted, click near the middle of the face to specify the center point of the circle.  
The UCS icon is positioned at the point picked, and its orientation is changed to match the face of the 3D solid.
5. Enter a radius of **0.125** for the circle and press Enter.  
The UCS icon returns to its previous location after the circle has been created.
6. Save the drawing with the name `Completed4-7.dwg` to the Chapter 4 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Answers to Practice Opportunities

### 1. How can you specify a coordinate value?

A, B, and D. You can specify coordinates in a number of ways, from manually entering values at the command line to picking points in the drawing using object snaps or using snap mode to specify points based on an incremental value. C is the incorrect answer because you use Snap mode to select points in the drawing based on an incremented value or invisible grid, not by picking points on an object.

### 2. How do you set running object snaps?

B, C, and D. Running object snaps are set by selecting the object snaps you want to use on the Object Snaps tab in the Drafting Settings dialog box. A is the incorrect answer because you can only specify an object snap override from the shortcut menu.

**3. When do you use the WCS, and when do you use the UCS?**

You always use the WCS when possible, but there are many cases when you want to draw objects on the surface of a 3D object, such as to create a hole or to add a new set of surfaces. When you want to draw on the surface of an object, you want to define a new UCS.

**4. Besides choosing View⇨Display⇨UCS Icon⇨On, how can you change where the UCS icon is displayed?**

B and C. You can display the UCS icon by choosing or View⇨Display⇨UCS Icon⇨Origin, or by using the UCSICON command. A is incorrect because no option is available from any short-cut menu that allows you to control the display of the UCS icon. D is incorrect as well because this menu choice doesn't exist.

## Chapter 5

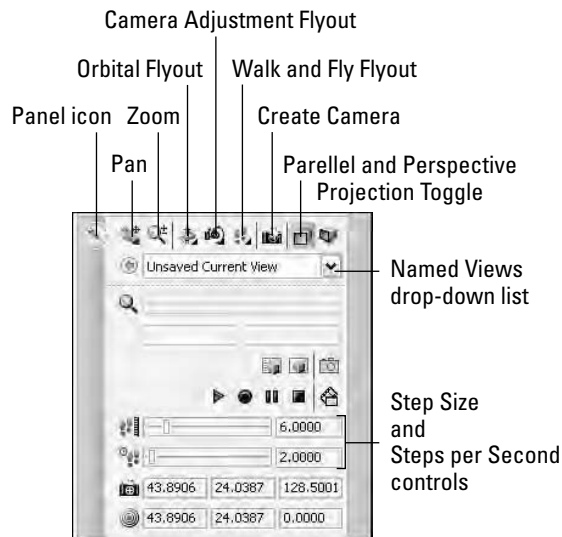
# Viewing a 3D Model

### *In This Chapter*

- ▶ Working with orthogonal and isometric views
- ▶ Panning, zooming, and orbiting in a 3D model
- ▶ Using named views, cameras, and backgrounds
- ▶ Enhancing the view of the 3D model with visual styles
- ▶ Controlling the quality of a 3D model's appearance

You can view a drawing in many different ways, but when you work in 2D, you're essentially limited to panning and zooming. While working in 3D, you have an additional axis that you must contend with. AutoCAD allows you to view your 3D models using orthogonal and isometric preset viewpoints, along with giving you the ability to adjust the view of a 3D model by panning, zooming, and orbiting (rotating/twisting a view). The exercises in this chapter give you an understanding of how to navigate around a 3D model and use some basic visualization methods.

When you're navigating 3D models, most of the common tools that you need exist nicely in one place. The 3D navigation tools that are introduced in this chapter are on the 3D Navigate panel (see Figure 5-1) of the Dashboard panel.



**Figure 5-1:**  
The 3D  
Navigate  
panel on the  
Dashboard.

## Establishing the Point of View

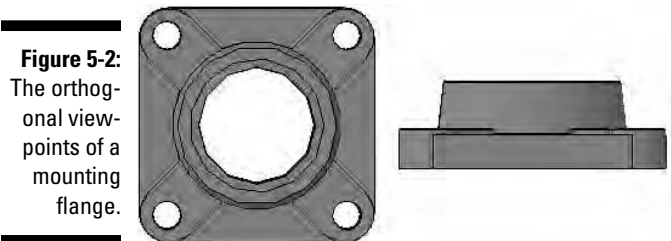
As you start working in 3D, you need to change the way you look at your 3D model. AutoCAD allows you to change the current viewpoint of a 3D model in a number of different ways. You can use preset viewpoints to make navigating your 3D models easier when modeling or use a custom viewpoint when plotting.

### Orthogonal viewpoints

AutoCAD comes with a number of orthogonal preset viewpoints that allow you to navigate around a 3D model. There are a total of six preset orthogonal viewpoints, and they are parallel to the current UCS (user coordinate system): Top, Bottom, Left, Right, Front, and Back. You specify orthogonal viewpoints by doing any of the following:

- ✓ Choose View↔3D Views and select a viewpoint.
- ✓ Select a preset viewpoint from the Named Views drop-down list on the Views toolbar or the 3D Navigate panel.
- ✓ Enter **-view** at the command prompt and select the Orthographic option.

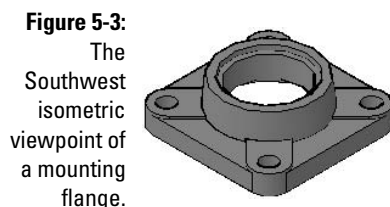
Figure 5-2 shows a 3D model mounting flange from a Top and Left viewpoint.



**Figure 5-2:**  
The orthogonal viewpoints of a mounting flange.

### Isometric viewpoints

AutoCAD comes with four preset isometric viewpoints that allow you to view a 3D model parallel to the current UCS and from above: Southwest, Northwest, Northeast, and Southeast. Isometric viewpoints allow you to view a model from above, and you specify them the same way you do orthogonal viewpoints. If you want to view the model from below in an isometric view, you use the Viewpoint Presets dialog box or one of the 3D orbiting commands that I talk about later in this chapter. I cover the Viewpoint Presets dialog box in the “Using the Viewpoint Presets dialog box” section and the 3D orbiting commands in the “Orbiting” section. Figure 5-3 shows a 3D model of a mounting flange from a Southwest viewpoint.



**Figure 5-3:**  
The Southwest isometric viewpoint of a mounting flange.

### Exercise 5-1: Using Orthogonal and Isometric Preset Viewpoints

For this exercise, open the drawing file `Exercise5-1.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice viewing the drawing using some of the preset views.

1. Display the Dashboard.

Choose Tools→Palettes→Dashboard.

2. On the Dashboard, select Top from the Named Views drop-down list on the 3D Navigate panel.

If the 3D Navigate panel isn't displayed on the Dashboard, right-click one of the displayed panels and choose Control Panels→3D Navigate.

The drawing switches to the Top preset view.

3. From the Named Views drop-down list, select Bottom.

The drawing switches to the Bottom preset view.

4. From the Named Views drop-down list, select Southwest.

The drawing switches to the Southwest preset view, which allows you to look at the drawing from above the part.

5. Close the drawing without saving the changes.

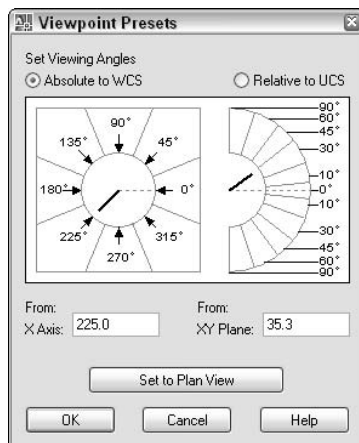


By default, AutoCAD uses smooth transitions to switch from one viewpoint to another, which helps you keep your orientation in the drawing. You can control the speed — frames per second — and which viewing functions use smooth transitioning with the View Transitions dialog box. To display the View Transitions dialog box, enter `vtoptions` at the command prompt.

## Using the Viewpoint Presets dialog box

Using orthogonal and isometric preset viewpoints is great to quickly navigate around your 3D model, but sometimes you want to view your 3D model from a nonpreset viewpoint, such as from the bottom of a 3D model at an isometric view. The Viewpoint Presets dialog box (see Figure 5-4) allows you to view a 3D model from a specific angle along the X axis and from the XY plane. As you specify a viewpoint, you can specify the angles in the world coordinate system (WCS) or in the current user coordinate system (UCS).

**Figure 5-4:**  
The Viewpoint Presets dialog box allows you to specify a custom viewpoint.



### Exercise 5-2: Changing the Viewpoint with the Viewport Presets Dialog Box

For this exercise, open the drawing file `Exercise5-2.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice changing the viewpoint of a 3D model with the Viewpoint Presets dialog box.

1. Choose View⇨3D Views⇨Viewpoint Presets.
2. In the Viewpoint Presets dialog box, click 135° on the left side of the image to specify the From X Axis value.
3. Click near 60° on the right side of the image to specify the From XY Plane value.
4. Click OK.

The viewpoint updates and shows a view that's similar to a Northwest isometric view, but using a greater angle to give the view a more top-down look at the model.

5. Display the Viewpoint Presets dialog box again; this time, enter the value **250** in the From X Axis text box and **-45** in the From XY Plane text box.
6. Click OK.

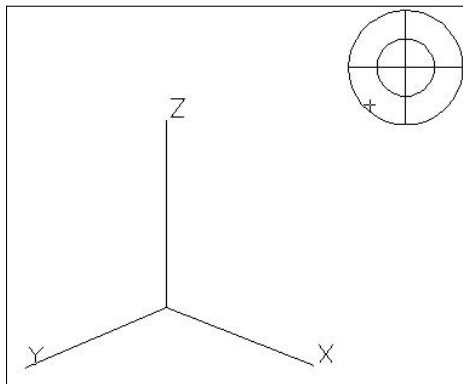
The viewpoint updates and shows an isometric view of the bottom of the 3D model.

7. Close the drawing without saving the changes.

## Accessing the tripod and compass to find your way

AutoCAD is a very visual application, and at times you want to interact with the drawing window directly to manipulate the current viewpoint instead of using one of the preset views or the Viewpoint Presets dialog box. The VPOINT command allows you to set the current viewpoint by interacting with what's known as the tripod and compass (see Figure 5-5). The tripod shows you the direction of the X, Y, and Z axes in the current UCS based on the location specified by the compass. The compass is located in the upper-right corner of the current viewport and is a 2D representation of a globe; the center point is the top (north pole), the outer ring is the bottom (south pole), and the inner ring is the equator. If you know the specific angle from the X axis and the angle from the XY plane, you can use the Rotate option of the VPOINT command.

**Figure 5-5:** When packing your AutoCAD 3D survival kit, don't forget the tripod and compass.



When working with 3D models, you spend a lot of time navigating. If you find yourself using a number of custom viewpoints, you might want to create a custom toolbar with buttons to set the current viewpoint for you.





1. To create a custom toolbar, choose Tools⇨\_\_\_\_\_⇨\_\_\_\_\_. Then, in the Interface dialog box, click the \_\_\_\_\_ tab.

### Exercise 5-3: Defining a Viewpoint with the Tripod and Compass

For this exercise, open the drawing file `Exercise5-3.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice changing the viewpoint of a 3D model with the tripod and compass.

1. Choose View⇨3D Views⇨Viewpoint.
2. In the drawing window, drag the cursor in the compass to the lower-right quadrant and click.

The viewpoint updates and shows an isometric view of the bottom of the mounting flange.

3. Close the drawing without saving the changes.



## Standard Viewing Mechanics

If you've done a fair amount of 2D drafting, you most likely are familiar with two of the main ways to get around and up close to your designs: panning and zooming. Along with panning and zooming in a 3D model, you can also adjust the distance from an object, and you can even swivel the current view as if you're standing in one spot and turning around that point, walking through or around a 3D model, or putting on a cape and flying through or around a 3D model.

### Zooming and panning

Panning and zooming in 3D are no different than they are in 2D with the exception that instead of using the PAN command you use 3DPAN, and you use 3DZOOM instead of ZOOM. Although it might seem somewhat confusing that AutoCAD uses different commands, you really don't have to worry too much about it because AutoCAD simplifies everything by using the 3D commands when a 3D visual style is active. I talk about visual styles later in this chapter, in the "Giving Your 3D Models Style (Fashion Tips Not Included)" section.

Table 5-1 lists the basic 3D navigation commands that you can use to navigate a 3D model.

<i>Button</i>	<i>Navigation Method (Command Name)</i>	<i>Description</i>
	Pan (3DPAN)	Allows you to interactively drag the current view in any direction parallel to the screen to see other parts of your 3D model without changing the current zoom magnification.
	Zoom (3DZOOM)	Allows you to interactively drag the current view to increase or decrease the magnification of the camera, making objects appear larger or smaller.

When the Pan or Zoom commands are active, you can access a number of 3D navigation options via the right-click menu. You can access other 3D navigation commands like Constrained Orbit and Swivel along with switching between parallel and perspective projection, among other options.



If you have an IntelliMouse or a mouse that has three buttons, you can perform these actions:

- ✓ **Pan in the current viewport:** Press and hold down the scroll wheel or the middle mouse button. To use the scroll wheel or middle button to pan, set the system variable MBUTTONPAN to 1 and be sure the mouse is properly configured. While holding down the scroll wheel or middle button, you can initiate orthogonally panning by holding the Shift key.
- ✓ **A zoom extents in the current viewport:** Double-click with the scroll wheel or the middle button.
- ✓ **Zoom in and out in real time:** Scroll the wheel forward and backward. The system variable ZOOMFACTOR controls the magnification percentage the drawing is zoomed in or out when scrolling the wheel on the mouse. The system variable ZOOMWHEEL controls in which direction the mouse wheel is scrolled to zoom in or out.



When a 3D navigate command is active like 3DPAN and 3DZOOM, switch between different 3D navigation commands by pressing the numeric key next to one of the options under the Other Navigation Modes submenu on the right-click menu.

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#### Exercise 5-4: Navigating a 3D Model Using the Basics

For this exercise, open the drawing file `Exercise5-4.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice using some of the basic 3D navigation commands and some of the options from the right-click menu when a 3D navigation command is active.

1. On the Dashboard, select Zoom from the 3D Navigate panel.
2. Click in the drawing window, and drag up and drag down.

Dragging up zooms in, and dragging down zooms out so you can see more of the drawing.

If you're using Windows XP, you can drag past the edge of the drawing window to keep zooming in or out if you need to.

3. With Zoom still as the active 3D navigation mode, press 9 on the keyboard to switch to panning.
4. Click in the drawing window and drag in any direction to pan the drawing.
5. Right-click the drawing window and choose Reset View.

The view is reset to the initial view that was set before you started zooming and panning in the drawing.

6. Right-click the drawing window and choose Zoom Extents.

The view is changed to show the extents of the model.

7. Press Esc or Enter to exit the current 3D navigation command.
8. Close the drawing without saving the changes.






## Orbiting

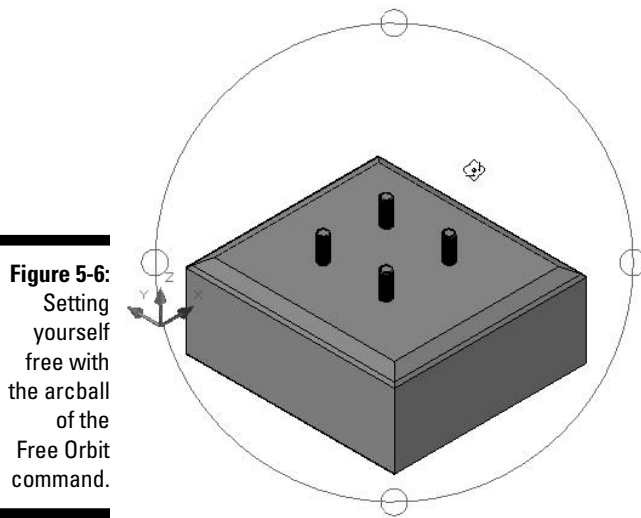
Outside of changing the viewpoint using one of the presets, panning, or zooming in on a 3D model, you can rotate your model interactively on-screen. The proper terminology of rotating a 3D model is *orbiting*. Orbiting a 3D model is very similar to the way the moon orbits around the Earth; in AutoCAD, your model remains in a fixed location while the current view is updated to reflect your changes. When using orbiting, you are navigating along a spherical path; the point you rotate around is different for each of the three orbiting commands that is in AutoCAD.

You can activate one of the orbit commands by doing any of the following:

- ✓ Choose View↔Orbit.
- ✓ Click a button from the 3D Navigation toolbar or the 3D Navigate panel on the Dashboard.
- ✓ Choose an option from the 3D Navigation Modes shortcut menu.
- ✓ Enter an orbit command at the command prompt.

Table 5-2 lists the different 3D orbit commands that you can use to navigation around a 3D model.

Table 5-2		3D Orbit Commands
Button	Orbit Type (Command Name)	Description
	Constrained (3DORBIT)	Allows you to orbit around a stationary point at the center of the objects in the current viewpoint when the command is started. You can adjust the current view parallel to the current XY plane by dragging horizontally and by dragging vertically to move the camera along the Z axis.
	Free (3DFORBIT)	Allows you to orbit around the center point of what is called the arcball. The arcball (see Figure 5-6) allows you to control the orbiting around a 3D model in four distinctly different ways. Clicking and dragging in the small circles located at the top and bottom of the arcball allow you to rotate the model around the horizontal axis of the arcball, and clicking and dragging in the small circles on the left and right sides of the arcball allow you to rotate the model around the vertical axis of the arcball. Dragging outside the arcball allows you to roll the current view perpendicular to the screen, and dragging inside the arcball allows you to freely rotate the model.
	Continuous (3DCORBIT)	Allows you to set the 3D model in motion on-screen based on the direction that you specify. The model's orbiting speed is determined by the distance that you cover when dragging in the direction to start the orbiting. Dragging to start the orbiting for the Continuous Orbit command is similar to dragging to start the orbiting for the Constrained Orbit command.



**Figure 5-6:**  
Setting  
yourself  
free with  
the arcball  
of the  
Free Orbit  
command.



When working with a large model, you can select some of the objects in the drawing before starting one of the 3D orbit commands. The objects that you select will be the only objects that appear on-screen while the orbit command is active; this also applies to other 3D navigation commands if they are started while one of the orbit commands is active.



If you have an IntelliMouse or a mouse that has three buttons, you can press and hold down the Shift key and the scroll wheel or the middle mouse button to start the Constrained Orbit command transparently in the current viewport.



### Exercise 5-5: Orbiting around a Model

For this exercise, open the drawing file `Exercise5-5.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice using the 3D orbit commands to navigation around a model.

1. Select the base in the drawing and not the flange.
2. On the Dashboard, select Constrained Orbit from the Orbit flyout on the 3D Navigate panel.  
The mounting flange disappears when the Constrained Orbit command starts.
3. Click and drag in the drawing window. Drag in a few different directions.  
As you drag in different directions, the 3D model seems to rotate around a fixed point.
4. Press 2 on the keyboard to switch to Free Orbit.  
The arcball is displayed in the center of the drawing window and the mounting flange remains invisible.
5. Click and drag inside the arcball. Drag in a few different directions.  
As you drag in different directions, notice that the 3D model doesn't rotate quite the same as it did with the Constrained Orbit command.
6. Click and drag outside of the arcball and in the different small circles located around the edge of the arcball.

Each one of these locations allows you to interactively update the current view differently.

7. Right-click the drawing window and choose Other Navigation Modes↔ Continuous Orbit.
8. Click and drag in any direction.  
The model begins to rotate on its own in the direction that you specified. The speed at which the model orbits is based on how fast you clicked and dragged in the drawing window.
9. Click the drawing window to stop the model from continuously orbiting.
10. Press Esc or Enter to exit the current 3D navigation command.
11. Close the drawing without saving the changes.

## Looking at a model from two different ways

AutoCAD allows you to view your 3D models in two different projections: parallel and perspective. When using parallel projection, you see all objects in the same depth no matter how close or far they are in the current view, which makes drawing much easier in 3D. However, in real life, you don't see objects in a parallel view but rather in perspective. In perspective, objects that appear closer to you look larger whereas objects farther away look smaller no matter if the objects are identical in size. Figure 5-7 shows a prototype of a level in both parallel and perspective projection. Perspective is available only when a 3D visual style is active. I talk about visual styles later in this chapter, in the section "Giving Your 3D Models Style (Fashion Tips Not Included)."



2. With 2D drawings, you use only one projection. Which one and why?

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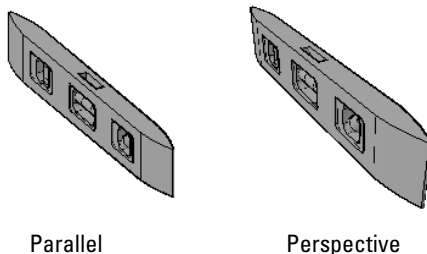


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**Figure 5-7:**  
A prototype  
of a level  
shown in  
both parallel  
and per-  
spective  
projection.



Parallel

Perspective

You can switch between parallel and perspective projection with the following methods:

- ✓ Select Parallel Projection or Perspective Projection on the 3D Navigate panel on the Dashboard.
- ✓ At the command prompt, enter **perspective** and enter a value of 0 to set parallel projection active or 1 to set perspective projection active in the current view.

### Exercise 5-6: Changing the Current Projection





For this exercise, open the drawing file `Exercise5-6.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice switching between parallel and perspective projection.

1. On the Dashboard, select Perspective Projection on the 3D Navigate panel.  
The view changes and the front edge of the 3D model closest to you appears taller than the back edge. You might also notice that the background color of the drawing window changes. This is because AutoCAD is now in the 3D perspective projection context.
2. On the Dashboard, select Parallel Projection on the 3D Navigate panel.  
Both the front and back edges of the 3D model appear to be the same height once again.
3. Close the drawing without saving the changes.

## Advanced Viewing Mechanics

Although panning, zooming, and orbiting are the most common 3D navigation methods in AutoCAD along with using preset viewpoints, you can use a few other advanced methods to navigate a 3D model. Other 3D navigation methods in AutoCAD allow you to adjust the position of the current camera as you look at a 3D model, and they give you the ability to walk/fly through a drawing — no superpowers or cape required.

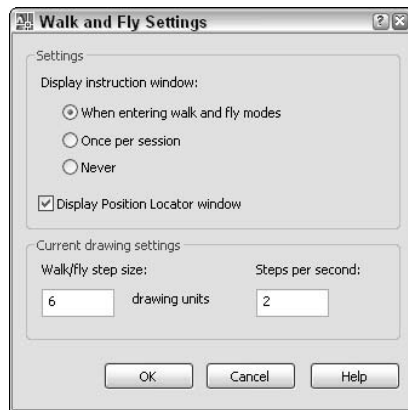
Table 5-3 lists the advanced 3D navigation commands that are available.

<b>Button</b>	<b>Navigation Method (Command Name)</b>	<b>Description</b>
	Distance (3DDISTANCE)	Allows you to interactively adjust the source of the camera so objects appear closer or farther away.
	Swivel (3DSWIVEL)	Allows you to change the target of the camera interactively. Swiveling gives you the opposite effect of orbiting; instead of rotating around the 3D model, you are turning in a circle, causing the 3D model to orbit around you.
	Fly (3DFLY)	Allows you to interactively adjust the current view of the drawing as if you were flying through it. While the command is active, the camera is free to move above the XY plane to give you the effect that you're lifting up into the air. As you move through the drawing, the Z height doesn't remain constant.
	Walk (3DWALK)	Allows you to interactively adjust the current view of the drawing as if you were walking up to it. Unlike the Fly command, the Walk command keeps you on the current XY plane to give you the effect that you're on the same plane as the model instead of floating above it. As you move through the drawing, the Z height remains constant.

## Configuring Fly and Walk

You can control the distance covered in the drawing when using the Walk and Fly commands by setting the step size and the number of steps per second. The more steps per second, the more distance you cover in a shorter period of time. Similarly, the larger the step size, the larger the distance you can cover in less time. However, the larger the step and the more steps you take, the harder it is to see your 3D model. To specify the step size and the steps per second, you can use the sliders/text boxes when the 3D Navigate panel of the Dashboard is expanded or use the Walk and Fly Settings dialog box (see Figure 5-8), which can be displayed from the Walk and Fly flyout on the 3D Navigate panel.

**Figure 5-8:** Adjust the settings for the Walk and Fly commands by using the Walk and Fly Settings dialog box.



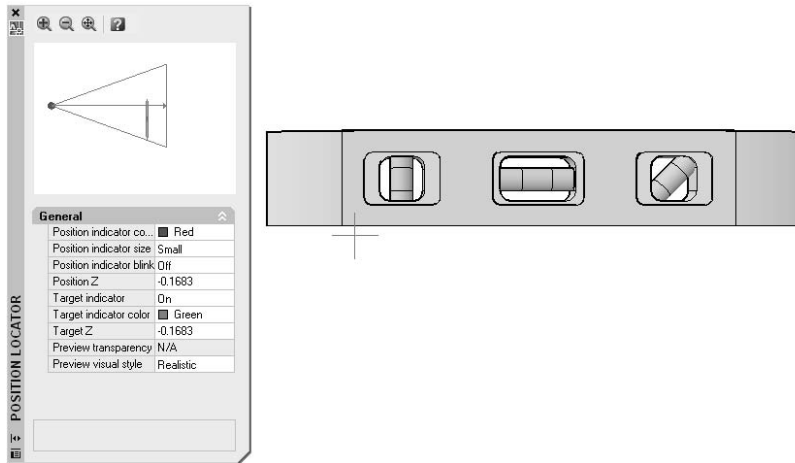
## Navigating with Fly and Walk

While navigating a model in the Fly and Walk modes, you use both the mouse and the keyboard to move and look around. You can use the following keys and mouse actions to navigate and switch between the Walk and Fly modes:

- ✓ **W or ↑**: Moves the camera forward.
- ✓ **A or ←**: Moves the camera to the left.
- ✓ **S or ↓**: Moves the camera backwards.
- ✓ **D or →**: Moves the camera to the right.
- ✓ **F**: Switches between Fly and Walk mode.
- ✓ **Left mouse button**: Swivels the camera and allows you to look around.

To use the Walk and Fly commands, you must be viewing the drawing with a perspective projection. (For information on how to toggle between parallel and perspective projections, see the section “Looking at a model from two different ways” earlier in this chapter.) After starting the Walk and Fly commands, the Position Locator window is displayed to help you navigate your 3D model. The Position Locator window (see Figure 5-9) provides you with a bird’s-eye of your model and gives you some different ways to navigate a model from the preview window.

**Figure 5-9:**  
The Position  
Locator  
window  
gives you a  
bird's-eye  
view of the  
3D model  
and your  
location.



### Exercise 5-7: Looking Around a Model



For this exercise, open the drawing file `Exercise5-7.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice using the advanced 3D navigation commands.



1. On the Dashboard, select Distance from the Camera Adjust flyout on the 3D Navigate panel.  
Click and hold down the left mouse button to make the flyout appear. Then drag down to the Distance button and release the mouse.
2. Click and drag in the drawing window; drag up to bring the 3D model closer or down to push it farther away.
3. On the Dashboard, select Swivel from the Camera Adjust flyout on the 3D Navigate panel.
4. Click and drag in the drawing window; drag up and down to adjust the camera's target along the Z axis or left and right to adjust the camera's target along the XY plane.



If you keep dragging in one direction, the camera eventually goes a full 360 degrees, returning the camera to the original target point.

5. Right-click the drawing window and choose Preset Views  $\rightarrow$  Left.
6. Press Esc or Enter to exit the Swivel navigation command.
7. On the Dashboard, select Walk and Fly Settings from the Fly and Walk flyout on the 3D Navigate panel.
8. In the Walk and Fly Settings dialog box, select Once Per Session for Display Instruction Window under the Settings, and make sure the Display Position Locator Window check box is selected.
9. Under Current Drawing Settings, set the Walk/Fly Step Size to 2 and Steps Per Second to 2.5.
10. Click OK to return to the drawing window.
11. On the Dashboard, select Walk from the Fly and Walk flyout on the 3D Navigate panel.



Because you're not in perspective projection, a Warning dialog box might display. Click Yes to continue.

12. In the Walk and Fly Navigation Mappings dialog box, click Close.

This dialog box serves as a reminder that the navigation controls are for the Fly and Walk commands. You can control whether the dialog box displays under the Display Instruction Window option in the Walk and Fly Settings dialog box. When the Walk and Fly Navigation Mappings dialog box is closed, the Position Locator window is displayed.

13. Press the W, S, A, or D key to walk up to the 3D model.

The target point of the camera is defined by the large green indicator in the middle of the drawing window. (You can turn off the indicator and change its color in the Position Locator window.)

14. Click in the drawing window and drag in any direction to adjust the target point of the camera, which allows you to look left, right, up, or down.

As you adjust your viewing angle, you can see the changes in the Position Locator window. Notice as you look up or down the PositionZ value doesn't change. That's because you're positioned on the current XY plane.



You can select the green target cone in the Position Locator window to pan, adjust the current view of the model along the XY plane, and adjust the distance to and from the 3D model based on where you click and drag on the green target cone. You can also use the Zoom In, Zoom Out, and Zoom Extents tools at the top of the Position Locator window to control how much of the 3D model you see in the Position Locator window's bird's-eye preview box.

15. Press F or 7 to switch to Fly mode.
16. Press the W, A, S, or D key to fly up to the 3D model. Just like Walk mode, you can adjust the target of the camera by clicking and dragging in the drawing window.

As you navigate to the drawing, the PositionZ value changes based on your current viewing angle of the 3D model. You have now left the XY plane, so you can zoom to a specific spot in the model.

17. Close the drawing without saving the changes.

The Distance, Swivel, Fly, and Walk commands are very useful if you want to create a quick animation through or around a 3D model using the animation feature that is in AutoCAD. Chapter 18 covers the animation features.

## Setting the Stage: Cameras and Backgrounds

Changing the viewpoint of a 3D model is a great way to access different sections of it. If you find yourself going back to a particular view a number of times, consider using named views or cameras. Named views and cameras are similar, with one slight difference: You can modify cameras to change the current view shown by the camera interactively using grips.

After you have a named view or camera created, you can assign it a background that appears in the current viewport when a 3D visual style is applied or a rendering is done. I talk about visual styles later in this chapter, in the "Giving Your 3D Models Style (Fashion Tips Not Included)" section, and Chapter 18 covers rendering. Backgrounds

give more realism to a rendering, such as the background of a factory floor where a machine of a 3D model might be installed. You can also add your company logo/watermark to a rendering.

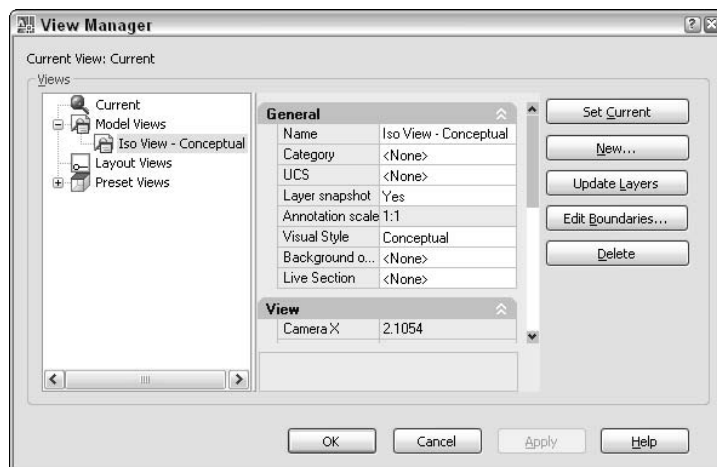
## Named views

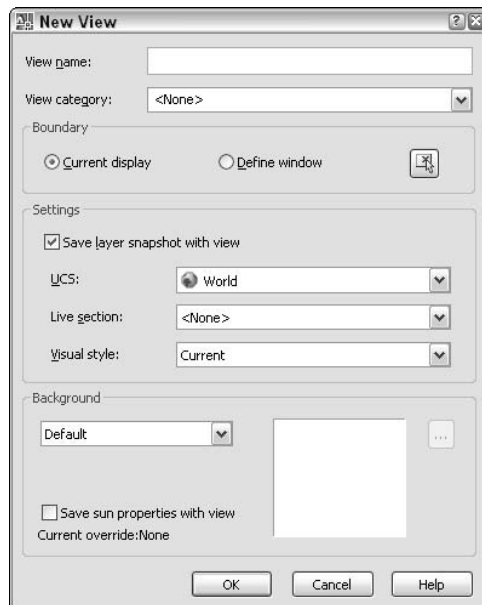
Named views have existed in AutoCAD for a long time, but they're often a forgotten feature. Named views — as their name implies — allow you to assign a view of a 3D model a unique name. You might create named views to help simplify navigating your 3D model or to help ensure that the model is plotted consistently each time. A named view can also simplify viewing a complex model by setting up the specific layers that should be displayed and the annotation scale that should be used for the annotation objects in the view. Named views can also be used with the Sheet Set Manager to help organize parts on layouts. After you create a named view, you can use it as one of the preset viewpoints when navigating a 3D model.

To see your named views, choose View⇒Named Views to open the View Manager dialog box (see Figure 5-10). Click the New button to create a named view in the New View dialog box (see Figure 5-11). You can assign your view a name, the part of the drawing that you want to display, and a number of additional properties:

- ✓ **Layer Snapshot:** Restores the visibility states of the layers in the drawing when the view is set current.
- ✓ **UCS:** Specifies which UCS should be set current when the view is set current. For more information on the UCS, see Chapter 4.
- ✓ **Live Section:** Specifies which live section should be set current when the view is set current. For more information on creating live sections, see Chapter 10.
- ✓ **Visual Style:** Specifies which visual style to use when the view is set current. For more information on visual styles, see the section “Giving Your Designs Style (Fashion Tips Not Included)” later in this chapter.
- ✓ **Background:** Specifies the type of background that should be used when the view is set current. For more information on using backgrounds, see the section “Backgrounds” later in this chapter.

**Figure 5-10:**  
The View Manager dialog box allows you to create and manage your named views.





**Figure 5-11:**  
Named views are initially defined in the New View dialog box.



### Exercise 5-8: Creating a Named View

For this exercise, open the drawing file `Exercise5-8.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice creating a named view.

1. On the Dashboard, select New View from the Named Views drop-down list on the 3D Navigate panel.
2. In the New View dialog box, enter **Iso View - Conceptual** in the View Name text box.
3. Select Define Window.

The New View dialog box closes, and you return to the drawing window.

4. Pick the first point for the view just above and to left of the 3D model, and pick the second point to the right and just below the 3D model.
5. Press Enter to accept the defined view and to return to the New View dialog box.
6. In the New View dialog box, select Conceptual from the Visual Style drop-down list.
7. Click OK to save the named view and return to the drawing window.
8. Save the drawing with the name `Completed5-8.dwg` to the Chapter 5 folder in the My Practice Drawings folder under My Documents and close the drawing.



### Exercise 5-9: Modifying a Named View

For this exercise, open the drawing file `Exercise5-9.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice modifying a named view.

1. On the Dashboard, select Manage Views from the Named Views drop-down list on the 3D Navigate panel.
2. In the View Manager dialog box, click the plus sign next to Model Views in the Views tree.

3. Select Iso View - Conceptual in the Views tree under the Model Views node and then click in one of the properties in the middle of the dialog box.
4. Select the UCS property to activate the drop-down list for the property.
5. From the UCS drop-down list, select World.
6. Scroll down in the Properties pane and locate the Perspective property.
7. Select the Perspective property to activate the drop-down list for the property.
8. From the Perspective drop-down list, select On.
9. Click OK to save the named view and return to the drawing window.
10. Save the drawing with the name `Completed5-9.dwg` to the Chapter 5 folder in the My Practice Drawings folder under My Documents and close the drawing.

You can set a named view current by choosing one from:

- ✓ The Views list in the View Manager dialog box.
- ✓ The Named Views drop-down list on the View toolbar or the 3D Navigate panel on the Dashboard.
- ✓ The Restore option from the `-VIEW` command.

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### Exercise 5-10: Setting a Named View Current

For this exercise, open the drawing file `Completed5-9.dwg` or `Exercise5-10.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice setting a named view current.

1. On the Dashboard, select Iso View - Conceptual from the Named Views drop-down list on the 3D Navigate panel.  
  
The view in the drawing changes to the settings saved with the Iso View - Conceptual view. The drawing now appears in a perspective view based on the defined window that you created, and the Conceptual visual style is applied.
2. Save the drawing with the name `Completed5-10.dwg` to the Chapter 5 folder in the My Practice Drawings folder under My Documents and close the drawing.

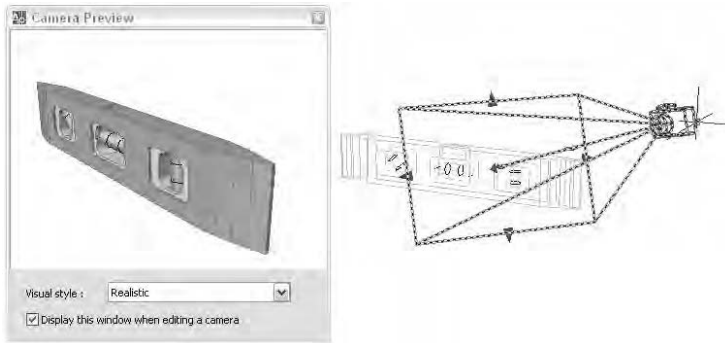
## Cameras

Cameras are similar to named views with one major difference: You can interact with cameras directly in the drawing window with what is known as a glyph. Camera glyphs allow you to adjust the source and target points of the camera and to adjust the lens length through the use of grips. As you modify a camera using its grips, the Camera Preview dialog box (see Figure 5-12) shows you a preview of what will be shown when the camera is set current. By default, camera glyphs are displayed in the drawing window and don't plot, but you can turn off glyphs to avoid accidental editing or erasing of cameras in the drawing, and you can set them to plot if you want.

You can create a camera by doing any of the following:

- ✓ Choose View↔Create Camera.
- ✓ Click Create Camera on the View toolbar or the 3D Navigate panel on the Dashboard.
- ✓ Enter **camera** at the command prompt.

**Figure 5-12:**  
While editing a camera, you can look through the camera with the Camera Preview dialog box.



AutoCAD provides access to a few different camera tools on the Cameras tool palette. To access these additional camera tools, click the icon in the upper-left corner of the 3D Navigate panel on the Dashboard.



### Exercise 5-11: Creating a Camera

For this exercise, open the drawing file `Exercise5-11.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice creating a camera.

1. Choose `View`↔`Zoom`↔`Extents`.
2. On the Dashboard, click `Create Camera` on the 3D Navigate panel.
3. At the prompt `Specify camera location:`, enter **13.5,-10,4** and press `Enter`.
4. At the prompt `Specify target location:`, enter **8,-17,1** and press `Enter`.
5. Enter **n** (for Name) and press `Enter`.
6. When prompted to enter a name for the camera, enter **Custom View** and press `Enter`.
7. Enter **v** (for View) and press `Enter`.
8. Enter **y** (for Yes) and press `Enter`.
9. Save the drawing with the name `Completed5-11.dwg` to the Chapter 5 folder in the My Practice Drawings folder under My Documents and close the drawing.

When a camera is selected, you can modify many of its properties other than just the source and target points and lens length. To edit the properties of a camera, you use the Properties palette. Along with interactively updating a camera in the viewport, you can use the View Manager dialog box to update the properties of a camera in the same way that you can modify a named view.



3. If you want to edit your camera's properties through the Properties palette, how do you display it?
  - A. Choose `Tools`↔`Palettes`↔`Properties`.
  - B. Click `Properties` on the Standard or Standard Annotation toolbar.
  - C. Choose `View`↔`Properties`.
  - D. Enter **properties** at the command prompt.
  - E. Click `Properties` on the Dashboard.



### Exercise 5-12: Modifying a Camera

For this exercise, open the drawing file `Completed5-11.dwg` or `Exercise5-12.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice modifying a camera.

1. Choose View↔Zoom↔Extents.
2. In the drawing window, select the Custom View camera.
 

A set of grips appears, allowing you to manipulate the target and source points along with the lens length of the camera. In addition to the grips, the Camera Preview dialog box displays. If the Camera Preview dialog box isn't displayed, right-click and select View Camera Preview.
3. In the Camera Preview dialog box, select Realistic from the Visual Style drop-down list.
 

The shading for the preview updates in the Camera Preview dialog box.
4. Select one of the arrow-shaped grips and drag it away from the target point. Click to adjust the lens length of the camera.
 

Dragging the arrow-shaped grips increases or decreases the lens length of the camera. As you adjust the lens length in the drawing window, the changes to the camera are shown in the Camera Preview dialog box.
5. Select the grip for the source point of the camera located inside the camera glyph and drag it in the drawing. Click to reposition the source point of the camera.
6. Right-click the drawing window and select Set Camera View.
7. Press Esc to deselect the camera glyph.
8. Save the drawing with the name `Completed5-12.dwg` to the Chapter 5 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Backgrounds

Backgrounds add another level of realism to your models when you're viewing, plotting, or rendering them. A background might be as basic as a solid color or something a little more elaborate that exists as an image. Backgrounds are a property of a named view or camera that can be changed when the view is initially created or later in the design cycle. The Background (see Figure 5-13) or Adjust Sun & Sky Background dialog box is displayed when assigning a background to a named view or camera.

**Figure 5-13:**  
The Background dialog box allows you to assign a solid color, a gradient color, or an image.



You can assign one of the following background types to a named view or camera:

<b>Background Option</b>	<b>Description</b>
Solid	Allows you to use a single color to fill the entire background of the viewport.
Gradient	Allows you to use two or three colors to fill the entire background of the viewport.
Image	Allows you to use a raster image that can be stretched, tiled, or centered, and you have some control over scaling and offsetting the image file in the viewport through the Adjust Background Image dialog box (see Figure 5-14).
Sun & Sky	Allows you to establish a representation of the sun and a background for the sky by using a number of different properties.



**Figure 5-14:** You have some control over how the background image is applied with the Adjust Background Image dialog box.

### Exercise 5-13: Assigning a Background with a Solid Color to a Camera

For this exercise, open the drawing file `Completed5-12.dwg` or `Exercise5-13.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice assigning a solid color background to a camera.

1. On the Dashboard, select Manage Views from the Named Views drop-down list on the 3D Navigate panel.
2. In the View Manager dialog box, click the plus sign next to Model Views in the Views tree.
3. Select Custom View in the Views tree under the Model Views node and then click in one of the properties in the middle of the dialog box.
4. Select the Background Override property to activate the drop-down list for the property.
5. From the Background Override drop-down list, select Solid.

6. In the Background dialog box, click the Color swatch under the Solid Options section.
7. In the Select Color dialog box, click the True Color tab and then pick a medium-blue color.
8. Click OK twice to return to the View Manager dialog box.
9. Click Set Current.
10. Click OK to return to the drawing window.  
The 3D model is displayed with the Conceptual visual style and a blue background.
11. Save the drawing with the name `Completed5-13.dwg` to the Chapter 5 folder in the My Practice Drawings folder under My Documents and close the drawing.



### Exercise 5-14: Assigning a Background with an Image to a Camera

For this exercise, open the drawing file `Completed5-13.dwg` or `Exercise5-14.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice assigning an image as the background for a camera.

1. On the Dashboard, select Manage Views from the Named Views drop-down list on the 3D Navigate panel.
2. In the View Manager dialog box, click the plus sign next to Model Views in the Views tree.
3. Select Custom View in the Views tree under the Model Views node and then click in one of the properties in the middle of the dialog box.
4. Select the Background Override property to activate the drop-down list for the property.
5. From the Background Override drop-down list, select Image.
6. In the Background dialog box, click Browse under the Image Options section.
7. In the Select File dialog box, browse to the Chapter 5 folder in the My Practice Drawings folder under My Documents and select `Clouds.jpg`. Click Open.
8. In the Background dialog box, click Adjust Image under the Image Options section.
9. In the Adjust Background Image dialog box, select Stretch from the Image Position drop-down list under the Background Preview section.
10. Click OK twice to return to the View Manager dialog box.
11. Click Set Current.
12. Click OK to return to the drawing window.

The 3D model is displayed with the Conceptual visual style and the image of the clouds that you assigned.

13. Save the drawing with the name `Completed5-14.dwg` to the Chapter 5 folder in the My Practice Drawings folder under My Documents and close the drawing.

## *Giving Your 3D Models Style (Fashion Tips Not Included)*

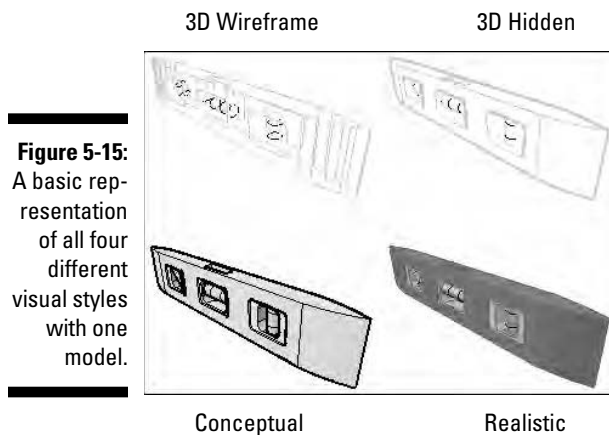
When you use different visual styles, the appearance of the 3D model changes. With visual styles, you can see what your model looks like in wireframe when model-



ing, and you can use a shaded view to see what the model might look like if it were manufactured. AutoCAD comes with five standard visual styles: One is for 2D drafting, and the other four are for 3D modeling. The four visual styles used for 3D modeling are

- ✔ **3D Wireframe:** Objects are displayed using lines and curves to represent the boundaries of 3D objects.
- ✔ **3D Hidden:** Outer edges of 3D objects are displayed with lines and curves, and back faces are hidden.
- ✔ **Conceptual:** Objects are shaded, and edges are given a smooth look between faces. Although the model isn't given a realistic appearance, the shading is done with transitions from warm to cool colors to make details of the 3D model easier to see.
- ✔ **Realistic:** Objects are shaded, and edges are given a smooth look between faces. The shading is done using the materials and colors assigned to the faces of the 3D model.

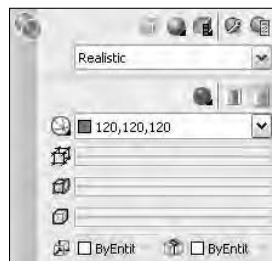
Figure 5-15 shows a 3D model with each of the four standard visual styles used for 3D modeling.



**Figure 5-15:** A basic representation of all four different visual styles with one model.

The Visual Styles panel (see Figure 5-16) of the Dashboard provides tools that allow you to assign a visual style to the current viewport and even tweak the current visual style in real time with some common properties for visual styles via sliders. To find out more about all the different settings that you can change for a visual style when creating or modifying one, see the *Customize a Visual Style* topic in the AutoCAD Online Help system. After you create a visual style, you can apply it to the current viewport or add it to a tool palette to use it in other drawings.

**Figure 5-16:** The Visual Styles panel on the Dashboard, in an expanded state.



You can set a visual style current in the current viewport by doing any of the following:

- ✓ Choose View⇨Visual Styles and select one of the standard visual styles.
- ✓ Select a visual style from the Visual Styles drop-down list on the Visual Styles panel of the Dashboard.
- ✓ Double-click or click the Apply button in the Visual Styles Manager palette.

### Exercise 5-15: Applying a Visual Style to the Current Viewport

For this exercise, open the drawing file `Completed5-14.dwg` or `Exercise5-15.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice assigning a visual style to the current viewport.

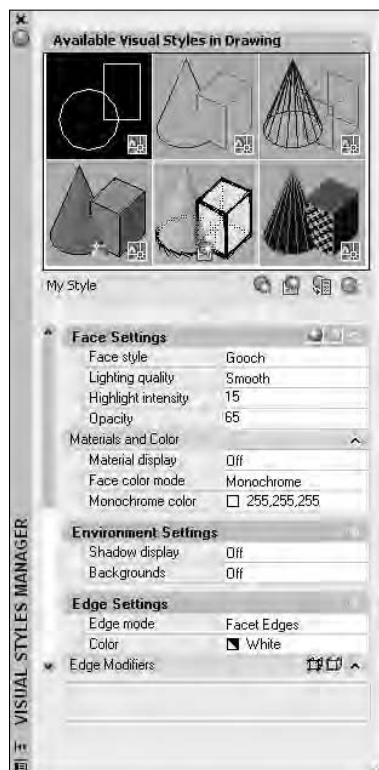
1. On the Dashboard, select Realistic from the Visual Styles drop-down list on the Visual Styles panel.

The shading of the 3D model changes.

2. Save the drawing with the name `Completed5-15.dwg` to the Chapter 5 folder in the My Practice Drawings folder under My Documents and close the drawing.

If you want more control over visual styles and the ability to use the same settings for a visual style a number of times and between drawings, you can create your own visual styles with the Visual Styles Manager palette (see Figure 5-17). You can find the palette by doing any of the following:

- ✓ Choose Tools⇨Palettes⇨Visual Styles.
- ✓ Click Manage Visual Styles on the Visual Styles toolbar or click Visual Styles Manager on the Visual Styles panel of the Dashboard.
- ✓ Enter **visualstyles** at the command prompt.



**Figure 5-17:**  
The Visual Styles Manager palette allows you to give your 3D models some flair.



AutoCAD provides some example visual styles that you can use with your 3D models and that can show you what you can do by changing the properties to create a custom visual style. These additional tools are on the Visual Styles tool palette that comes with AutoCAD. To access these additional visual styles, click the icon that is in the upper-left corner of the Visual Styles panel on the Dashboard.



### Exercise 5-16: Creating a Custom Visual Style

For this exercise, open the drawing file `Completed5-15.dwg` or `Exercise5-16.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice creating a new visual style by copying one of the standard 3D visual styles and making some changes to its properties.

1. On the Dashboard, click Visual Styles on the Visual Styles panel.
2. On the Visual Styles Manager palette, right-click the Conceptual visual style and select Copy.
3. Right-click the Conceptual visual style and choose Paste.  
A new visual style named Copy 1 of Conceptual is created.
4. Right-click the Copy 1 of Conceptual visual style and choose Edit Name and Description.
5. In the Edit Name and Description dialog box, enter **My Style** in the Name text box and **My custom style based on Conceptual** in the Description text box.
6. Click OK to return to the Visual Styles Manager palette.
7. Double-click the My Style visual style that you just created to set it current.
8. On the Face Settings caption, click Highlight Intensity (blue ball icon) and Opacity (transparent box with a sphere icon) located to the right of the caption.
9. Enter **15** for the value of the Highlight Intensity property and **65** for the value of the Opacity property.
10. Under Materials and Color, select Monochrome from the drop-down list for the Face Color Mode property.
11. Under Environment Settings, select Off from the drop-down list for the Backgrounds property.
12. On the Edge Modifiers caption, click Jitter Edges located to the right of the caption.  
The new visual style that you created gives the 3D model a sketched look that shows not only the front, but the back of the model as well because of the opacity value used.
13. Save the drawing with the name `Completed5-16.dwg` to the Chapter 5 folder in the My Practice Drawings folder under My Documents and close the drawing.

### Exercise 5-17: Adding a Visual Style to a Tool Palette

For this exercise, open the drawing file `Completed5-16.dwg` or `Exercise5-17a.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice creating a new tool palette so you can use a custom visual style in other drawings.

1. On the Dashboard, click Visual Styles on the Visual Styles panel.
2. Choose Tools⇨Palettes⇨Tool Palettes.

3. If the Tool Palettes window is docked, right-click the gripper bars located at the top of the palette. If the Tool Palette window is floating, right-click the title bar. Select the Visual Styles tool palette group.

The Visual Styles tool palette group is set current and by default has one tab on it.

4. Right-click an empty area of the tool palette and choose New Palette.
5. In the in-place editor, enter the name **My Visual Styles** for the name of the tool palette.
6. Click and drag the My Style visual style from the Visual Styles Manager palette and drop it onto the new tool palette.
7. Open the drawing `Exercise5-17b.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents.
8. With the `Exercise5-17b.dwg` active, click the My Style tool on the My Visual Styles tool palette.

The My Style visual style is applied to the current viewport of the drawing.

9. Save the drawing with the name `Completed5-17.dwg` to the Chapter 5 folder in the My Practice Drawings folder under My Documents and close the drawing.
10. Close the drawing without saving any changes.



You can remove unused visual styles using the Visual Styles Manager palette or the PURGE command. Keeping a drawing free of unnecessary information can help to reduce confusion on what is being used in the drawing and to help keep the file size of your drawings down.

## Controlling the Display Quality of 3D Solids

AutoCAD is optimized to provide a balance of creating accurate designs and to display a close representation of objects in the drawing window. Curved 2D objects and 3D solids can appear in the drawing window with jagged edges at times. The jagged edges are a result of AutoCAD displaying curved objects with short lines or vectors instead of a calculated arc or circle. By displaying curved objects with short lines, AutoCAD updates the display much faster by reducing the accuracy of the way the drawing is displayed, but this doesn't affect the actual accuracy of the objects in the drawing.

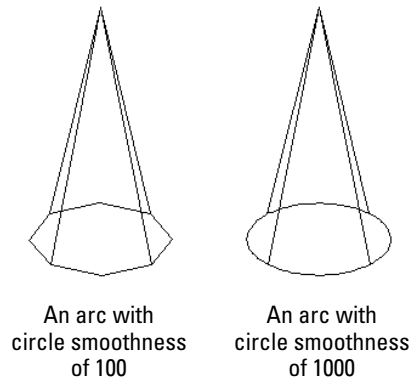


4. Identify which are curved 2D objects and which are 3D solids that can appear jagged.

<i>Object</i>	<i>2D Object or 3D Solid?</i>
Circle	_____
Arc	_____
Cone	_____
Cylinder	_____
Sphere	_____
Spline	_____
Torus	_____
Ellipse	_____

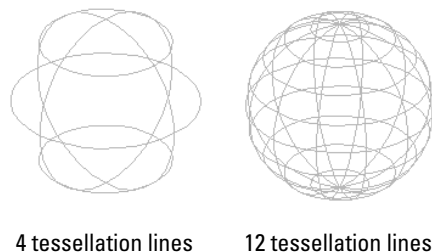
The Arc and Circle Smoothness setting on the Display tab in the Options dialog box controls how smooth 2D and 3D curved objects in the current viewport appear. A low value helps increase the performance of viewing large drawings in AutoCAD because it decreases the complexity of how objects appear on-screen. Figure 5-18 shows a cone displayed with the 3D Wireframe visual style and two different factors to control the smoothness of the object. The cone on the left is displayed using an arc and circle smoothness of 10, and the one on the right is displayed with an arc and circle smoothness of 1000 (which is the default drawing). An arc and circle smoothness of 100 is a nice number to help show the accuracy of a curved 3D and to increase the performance of AutoCAD when zooming in for large drawings.

**Figure 5-18:**  
Viewing curved objects with two different arc and circle smoothness factors.



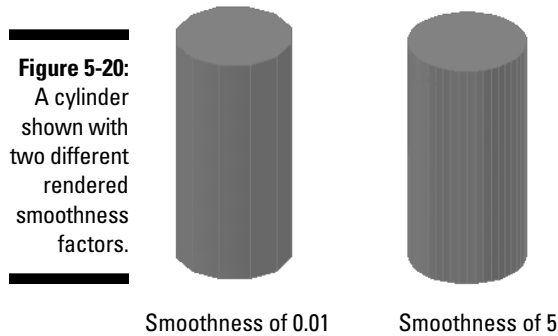
As you look at 3D solids with curves using either the 2D Wireframe or 3D Wireframe visual style current, you see tessellation lines. Tessellation lines show the curved features of a 3D solid when being viewed in wireframe. You can control the number of tessellation lines that are displayed by changing the value of Contour Lines per Surface on the Display tab of the Options dialog box. You can also set the number of tessellation lines that display a curved 3D object using the Number of Lines property for a 3D visual style. You can only set the number of tessellation lines once in AutoCAD; it's not a setting that can differ per object or viewport. Figure 5-19 shows a sphere (the one on the left) with 4 tessellation lines that run longitudinal to show the curve profile of the 3D solid, and the same sphere is shown on the right with 12 tessellation lines.

**Figure 5-19:**  
A sphere shown with two different sets of tessellation lines.



Another setting that can affect how curved 3D solids might appear in the current viewport when the 3D Hidden, Conceptual, or Realistic visual style is current is the smoothness of how shaded and rendered objects appear. The smoothness of a shaded or rendered 3D solid is controlled by the Rendered Object Smoothness field in the Options dialog box on the Display tab. A high value in the Rendered Object Smoothness field makes the faces for a curved 3D solid to appear smoother, while a low value makes the

faces for a curved 3D solid appear blocky or jagged. The smoothness range is 0.01 to 10, and by default it's set to 0.5. Figure 5-20 shows two cylinders displayed with two different smoothness factors: The one on the left is shown with a smoothness of 0.01, and the one on the right is shown with a factor of 5.



### Exercise 5-18: Changing the Display Quality of Curved 3D Solids

For this exercise, open the drawing file `Exercise5-18.dwg` from the Chapter 5 folder in the My Practice Drawings folder under My Documents. Then practice adjusting some of the settings that can affect the way curved 3D solids might appear in a viewport.

1. Display the Options dialog box and click the Display tab.
2. On the Display tab, under the Display Resolution section, enter **100** in the Arc and Circle Smoothness text box.
3. Click OK.

The sphere now appears curved instead of jagged.

4. Display the Visual Styles Manager palette and set the 3D Wireframe visual style current.
5. In the Visual Styles Manager palette, click the Number of Lines property under the Edge Settings section and enter **12** in the text box.

You can use the up and down spinners at the right side of the text box to change the number instead of typing one.

You can also change the value of the Contour Lines Per Surface text box in the Options dialog box. Changing the value in the Options dialog box doesn't change the visual style though; to use the same value each time, the visual style is set current.

6. Choose `View↔Regen` to regenerate the current viewport or `View↔RegenAll` to regenerate all the viewports in the entire drawing.

You should see the curves of the sphere better now than when only four tessellation lines were used to represent the sphere.

7. Choose `View↔3D Views↔SW Isometric`.
8. Set the Realistic visual style current.

The edges of the sphere appear jagged, and gaps appear between the tessellation lines and the shading of the sphere.

9. Display the Options dialog box and click the Display tab.

10. On the Display tab, under the Display Resolution section, enter 5 in the Rendered Object Smoothness text box.
11. Click OK.  
The shading of the sphere appears to follow the tessellation lines now.
12. Save the drawing with the name `Completed5-18.dwg` to the Chapter 5 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Answers to Practice Opportunities

1. To create a custom toolbar, choose **Tools**→**Customize**→**Interface**. Then, in the Interface dialog box, click the **Customize** tab.

For more information on customized toolbars, see Chapter 3.

2. With 2D drawings, you use only one projection. Which one and why?

Parallel projection. In perspective projection, objects that appear closer to you look larger, whereas objects farther away look smaller no matter if the objects are identical in size. Using parallel projection in a 2D drawing, which has no depth, makes no sense.

3. If you want to edit your camera's properties through the Properties palette, how do you display it?

A, B, and D. You can display the Properties palette in these ways:

- Choose **Tools**→**Palettes**→**Properties**.
- Click **Properties** on the Standard or Standard Annotation toolbar.
- Enter **properties** at the command prompt.

4. Identify which are curved 2D objects and which are 3D solids that can appear jagged.

<i>2D Objects</i>	<i>3D Solids</i>
Circle	Sphere
Arc	Cone
Spline	Cylinder
Ellipse	Torus





## Chapter 6

# Taking the Plunge into 3D

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### *In This Chapter*

- ▶ Setting up a drawing for success
  - ▶ Using helixes and polylines in the third dimension
  - ▶ Modeling with regions
  - ▶ Working with 3D modify commands
- 

**W**hen working in 3D, you have to figure out many new techniques, from entering coordinate values to navigating around a 3D model. After you feel comfortable with entering coordinate values and navigating in 3D, you're ready to create 3D objects. Do not fear; you can leverage what you already know about 2D drafting to create 3D models. The exercises in this chapter help you set up a new drawing that you can create a 3D model from, and I explain some of the basic 3D objects that you can create. Along with some basic modeling techniques, you discover how to use some of the basic 3D modifying commands.

## *Getting Started: Creating Your First 3D Model*

Creating a new drawing for 3D modeling isn't too much different from a drawing that you might create for drafting in 2D, but take these key differences into consideration when setting up your drawings for creating a 3D model:

- ✓ **Drawing units:** Drawing units not only specify how you input distances and angles, but also affect the scale in which blocks are inserted into a drawing and whether lighting is set to standard units or one of the available photometric unit values. For more information on lights and lighting units, see Chapter 16.
- ✓ **Grid and snap settings:** The grid is a useful visualization tool in 2D but is more helpful in 3D because of how it's implemented. The grid in 3D looks like a sheet of graph paper with major and minor lines, and it gives you a visual reference between distance and size. Snap settings work with the grid or with polar tracking to help you draw in specific increments. For more information on setting up the grip and snap spacing for use with 3D modeling, see the "Grid and snap" section, later in this chapter. Polar tracking and PolarSnap are also covered under the "Grid and snap" section, later in this chapter.
- ✓ **Limits:** Drawing limits allow you to control how large the grid is when you zoom in a drawing. Although establishing the drawing limits is nice, it's optional because you can set the grid to adapt to the current size of the drawing window. For more information on setting up drawing limits, see the "Limits" section, later in this chapter.
- ✓ **Object snaps and object snap tracking:** Although both object snaps and object snap tracking aren't stored with a drawing, you need to properly set up the features to ensure that you accurately create your models. For more information on object snaps and object snap tracking, see Chapter 4.

- ✓ **Drawing display:** When modeling, make sure you're working in parallel projection and the 3D Wireframe visual style to make snapping to objects easier. For more information on visual styles and drawing projections, see Chapter 5.
- ✓ **Layers:** Layers organize objects in a drawing and control the visibility of the objects on a layer. Layers are helpful when working in 3D not only to help organize parts for the model, but also to help separate construction geometry from the final modeled parts. To help identify construction geometry, you might want to create a layer called 3DConst or use another layer name that might be part of your layer standards. To make sure that the objects on a construction layer don't get printed, mark the layer as a nonplotting layer or freeze it.
- ✓ **Layouts:** If you want to use the Sheet Set Manager to help organize your 3D models or want to plot different views of your model, create layout tabs for the different views. For more information on layouts and plotting, see Chapter 20.
- ✓ **Modeling settings:** If you plan on using hardware acceleration or want to have the crosshairs or other interface elements look a specific way when you're modeling, you want to make the desired changes in the Options dialog box. For more information on setting up the options related to 3D modeling, see Chapter 3.
- ✓ **Establish the workspaces:** Using the 3D Modeling workspace that comes with AutoCAD, you can access many of the commonly used 3D-related commands. For more information on working with workspaces, see Chapter 3.



Although named objects like dimension styles, text styles, and table styles are important for 2D drafting, they aren't as important when creating a 3D model. However, if you use features like Flatshot or Livesection to create 2D drawings from your 3D models, make sure these styles are properly set up based on your company standards.

AutoCAD comes with two drawing template files that are designed with the focus of 3D modeling. The two drawing template files (`acad3D.dwt` for imperial drawings and `acadiso3D.dwt` for metric drawings) are very basic in what settings are established, but they provide a basic starting point with the setup of the grid, visual style, and drawing projection mode.



Take one of the templates that you might use for your 2D drafting and modify it for 3D modeling so you have all your existing text, dimension, and table styles as well as things like drawing units and snap settings.

For more information on working with drawing templates, see Chapter 2.

## *Grid and snap*

In many applications, close might be good enough, but when you create models in AutoCAD, you want them to be accurate because the drawings that are created from AutoCAD are used to manufacture parts with very close tolerances. AutoCAD provides many settings that help to ensure that models you create are accurate. One of the features in AutoCAD that can help you create accurate models is grid and snap.

Grid is a drafting aid that allows you to create a virtual sheet of graph paper to help you determine the distance you are creating an object at, whereas snap is used to help control the incremental distance that you move the crosshairs in across the screen. You modify the grid and snap settings using the Drafting Settings dialog box (see Figure 6-1). To display the Drafting Settings dialog box, choose **Tools**⇨**Drafting Settings**. With the Drafting Settings dialog box displayed, click the **Grid On** option to enable the use of the grid and **Snap On** to enable the use of snap.

**Figure 6-1:**  
Manipulating  
grid and  
snap with  
the Drafting  
Settings  
dialog box.



These are some of the properties that control how the grid works in 3D:

- ✔ **Major Line Every:** Major lines are displayed as bold or heavier lines on the grid to help you better estimate the distance you're moving along the grid.
- ✔ **Adaptive Grid:** The adaptive grid behavior ensures that the grid is displayed no matter the current zoom factor in the drawing. The grid lines that are displayed don't necessarily represent the actual spacing distance set for the drawing, though.
- ✔ **Allow Subdivision Below Grid Spacing:** Controls whether grid lines are displayed if you zoom in so close that the grid lines might be too far apart to be displayed in the current view.
- ✔ **Display Grid Beyond Limits:** Specifies whether the grid is displayed only at the size the current drawing limits are set to or whether the grid can be displayed beyond the specified drawing limits.
- ✔ **Follow Dynamic UCS:** If you use Dynamic UCS, this setting determines whether the grid is relocated on the face of the 3D solid that the UCS is temporarily associated with.

The grid in the drawing when a 3D visual style is applied is shown as a series of parallel and perpendicular lines, but if a 2D visual style is applied, the grid is displayed as a series of dots. By default, the behavior of the grid displays a major line for every fifth minor grid line. The major grid lines are of a heavier lineweight than the minor lines, making them appear bolder.

In combination with using the grid and snap to create accurate models, you can use ortho mode and polar tracking to restrict the angle from the previous point that you picked. Ortho mode restricts the crosshairs movement in 90-degree increments along the XY plane and in the -Z or +Z direction. Polar tracking restricts the crosshairs movement in a user-defined increment along the XY plane and in the -Z or +Z direction. To enable ortho mode, click the ORTHO button on the status bar or press the F8 key, and to enable polar tracking, click the POLAR button on the status bar or press the F10 key. Ortho mode and polar tracking are mutually exclusive features, meaning that one or the other is enabled but not both.

You define the angle for polar tracking on the Polar Tracking tab of the Drafting Settings dialog box. On the Polar Tracking tab, select an angle to use from the Increment Angle drop-down list or add one to the Additional Angles list. When using polar tracking, you can use the PolarSnap feature to specify an increment that the crosshairs can move in. To enable PolarSnap, display the Drafting Settings dialog box and click the Snap and Grid tab. On the Snap and Grid tab, select PolarSnap under the Snap Type section. When PolarSnap is enabled, enter the distance you want the crosshairs to move at in the Polar Distance text box under the Polar Spacing section.

To enable the grid, click GRID on the status bar, right-click GRID on the status bar and choose On, or enter **grid** at the command prompt and enter the option **on** or **off**. To toggle snap on and off, click SNAP on the status bar, right-click SNAP on the status bar and choose either PolarSnap On or Grid Snap On, or enter **snap** at the command prompt and enter the option **on** or **off**.

---

### Exercise 6-1: Adjusting the Grid and Snap

Practice creating a new drawing based on one of the drawing templates that comes with AutoCAD and make changes to both the grid and snap settings in the new drawing.

1. Choose File→New.
2. In the Select Template dialog box, select the `acadiso3D.dwt` drawing template file and click Open.
3. Right-click the GRID button on the status bar and choose Settings.
4. In the Drafting Settings dialog box, select the two options Snap On and Grid On along the top of the Snap and Grid tab.
5. In the Snap Spacing section, make sure Equal X and Y spacing is selected and then enter **1** in the Snap X spacing text box.
6. In the Grid Spacing section, enter **1** in the Grid X spacing text box and enter **10** in the Major Line Every text box.
7. In the Snap Type section, select Grid Snap and Rectangular Snap.
8. In the Grid Behavior section, make sure Adaptive Grid, Display Grid Beyond Limits, and Follow Dynamic UCS are all checked.
9. Save the drawing with the name `Completed6-1.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Limits

Drawing limits help control the size of the grid for the Model tab and a layout tab, but the setting isn't frequently used any more since the introduction of the Adaptive Grid and Display Grid Beyond Limits options in the Drafting Settings dialog box. Drawing limits are more commonly used for making sure that your 3D model or 2D drawing can be plotted on a specific sheet size.

You can change the current drawing limits by choosing Format→Drawing Limits or entering **limits** at the command prompt.

---

### Exercise 6-2: Adjusting the Limits of a Drawing

For this exercise, open the drawing file `Exercise6-2.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Then practice setting the drawing limits.

1. Choose Format⇨Drawing Limits.
2. At the Specify lower left corner or [ON/OFF] <0.0000,0.0000>: prompt, press Enter to accept the default location of 0,0.
3. At the Specify upper right corner <420.0000,297.0000>: prompt, enter **840,594** and press Enter.
4. Right-click the GRID button on the status bar and choose Settings.
5. In the Drafting Settings dialog box, on the Snap and Grid tab, deselect the Display Grid Beyond Limits option and click OK.
6. Choose View⇨Zoom⇨All.  
The grid is displayed as a rectangle now in the drawing and is the size of the specified drawing limits. The grid displayed this way gives you a reference of the overall size, and you can ensure that you can plot the 3D model on a specific sheet size from the Model tab.
7. Right-click the GRID button on the status bar and choose Settings.
8. In the Drafting Settings dialog box, on the Snap and Grid tab, select the Display Grid Beyond Limits option and click OK.
9. Choose View⇨Zoom⇨All.
10. Save the drawing with the name `Completed6-2.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

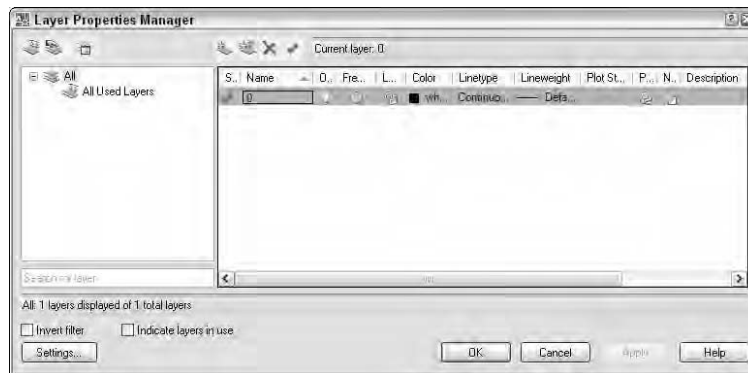
## Layers

Layers are a very powerful feature in AutoCAD, and you should use them consistently throughout each and every drawing that you create. Layers allow you to organize objects in a drawing that are used to communicate a feature or like parts of a design. Organizing layers allow you to make sure that groups of objects have the same entity properties, like color, linetype, lineweight, and plot style. Layers also allow you to lock selecting of objects and control the display of objects in the drawing window and whether the objects on the layer can be plotted. The Layer Properties Manager dialog box (see Figure 6-2) allows you to manage, create, and set a layer current.



When working with 3D, you want to make sure that your construction geometry is placed on a separate layer so it doesn't get plotted or lost when you create complex objects.

**Figure 6-2:**  
You use the Layer Properties Manager to create and manage the layers in a drawing.



You can create layers by doing any of the following:

- ✓ Choose Format⇨Layer.
- ✓ Click Layer Properties Manager on the Layers toolbar or the Layers panel on the Dashboard.
- ✓ Enter **layer** at the command prompt.



1. To find which layer is current, look at the current value displayed on the \_\_\_\_\_ drop-down list on the Layers toolbar or enter \_\_\_\_\_ at the command prompt.

To set a layer current, you can do one of the following:

- ✓ Make sure no objects are selected and select a layer from the Layer control on the Layers toolbar or the Layers panel on the Dashboard.
- ✓ Select an object on the layer you want to set current and click the Make Object's Layer Current button on the Layers toolbar or the Layers panel on the Dashboard.



To change the current layer of an object (or objects), make sure no command is active and select the objects in the drawing using a crossing window or window selection method and then select the Layer to set the selected objects to from the Layer control on the Layers toolbar or the Layers panel on the Dashboard.

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### Exercise 6-3: Creating New Layers for Modeling

For this exercise, open either the drawing file `Completed6-2.dwg` or `Exercise6-3.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Document. Then practice creating a few new layers in a drawing that will be used for construction geometry and the final modeled part.

1. Choose Format⇨Layer.
2. Click the New Layer button.
 

The New Layer button is located above the list of layers and looks like a sheet of paper with a yellow star above it.
3. Enter **Parts** for the layer name and enter , (a comma) to create a second layer.
4. For the second layer name, enter **3DConst** and press Enter.
5. Click the Color field for the layer Parts.
6. In the Select Color dialog box, click the Index Color tab and select color 5 (blue). Click OK.
7. Set the color of the layer 3DConst to color 7 (white).
8. Click the printer icon in the Plot field for the 3DConst layer to make sure that the objects on the layer aren't plotted.
9. Click OK to save the newly created layers.
10. Save the drawing with the name `Completed6-3.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Blurring the Lines between 2D and 3D

Until recently, AutoCAD has always been primarily looked upon as a 2D drafting package first and a 3D modeling application second. The approach that AutoCAD takes on 3D modeling is much different from an application like Autodesk Inventor that is designed from the ground up as a modeling program. Many of the objects that you might already be familiar with, like arcs and circles, in AutoCAD are 2D objects. If you want to create a circle in 3D space that is drawn at an angle, you need to adjust the UCS before drawing the circle; otherwise, it's a drawing on the current plane.

In addition to drawing lines with different start and end points and adjusting the UCS to draw 2D objects in 3D space, AutoCAD provides objects and methods that allow you to take your existing 2D geometry and create 3D objects from them or to draw true 3D objects in 3D space you can use to create wireframe models. In this section, you discover how to apply what you know about creating 2D objects to create 3D objects such as 3D polylines, helices, and extruding 2D objects.

### 3D polyline

AutoCAD allows you to create two different polylines (unless you count that there are two different 2D polylines, in which case you can create three different polylines). You can create a 2D and a 3D polyline that are different from each other. All the points (or vertexes) of a 2D polyline have the same Z coordinate value unless you draw it by using a custom UCS to achieve the appearance of it being drawn with multiple different Z values. You create 3D polylines, on the other hand, with varying Z coordinate values, but they're limited to straight segments only and can't have a width assigned to them. 3D polylines are somewhat limited as objects because they lack curves and width.

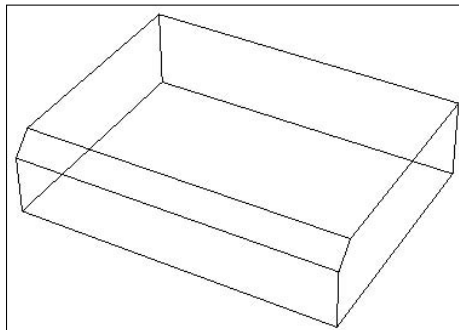


2. Why would you want to use a 3D polyline?

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You create 3D polylines by using the 3DPOLY command, and you edit them by using grips, Properties palettes, or the PEDIT command. Figure 6-3 shows a wireframe model of a part created using a 3D polyline.



**Figure 6-3:**  
Wireframe  
model cre-  
ated using  
3D polylines  
and lines.

---

**Exercise 6-4: Creating a Wireframe Model Using 3D Polylines and Lines**

For this exercise, open the drawing file `Completed6-3.dwg` or `Exercise6-4.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Then practice creating a wireframe model using 3D polylines and lines.

1. Click the DYN button on the status bar to turn off dynamic input.  
When the button is popped out, dynamic input is turned off.
2. From the Layer control on the Layers toolbar or the Layers panel on the Dashboard, select the layer 3DConst.
3. Choose Draw↔3D Polyline.
4. At the Specify start point of polyline: prompt, enter **0,0,0** and press Enter.
5. At the Specify endpoint of line or [Undo]: prompt, enter **6,0,0** and press Enter.
6. At the Specify endpoint of line or [Undo]: prompt, enter **6,0,2** and press Enter.
7. At the Specify endpoint of line or [Close/Undo]: prompt, enter **0.5,0,2** and press Enter.
8. At the Specify endpoint of line or [Close/Undo]: prompt, enter **0,0,1.5** and press Enter.
9. At the Specify endpoint of line or [Close/Undo]: prompt, enter **C** and press Enter.
10. Choose Modify↔Copy.
11. At the Select objects: prompt, enter **L** and press Enter twice.
12. At the Specify base point or [Displacement/mOde] <Displacement>: prompt, enter **0,8,0** and press Enter twice. If prompted to make another copy of the object, press Enter again.
13. Choose Draw↔Line.
14. At the Specify first point: prompt, enter **0,0,0** and press Enter.
15. At the Specify next point or [Undo]: prompt, enter **0,8,0** and press Enter twice.
16. Repeat the LINE command and draw a line from 6,0,0 to 6,8,0.
17. Repeat the LINE command and draw a line from 0,0,1.5 to 0,8,1.5.
18. Repeat the LINE command and draw a line from 0.5,0,2 to 0.5,8,2.
19. Repeat the LINE command and draw a line from 6,0,2 to 6,8,2.
20. Save the drawing with the name `Completed6-4.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

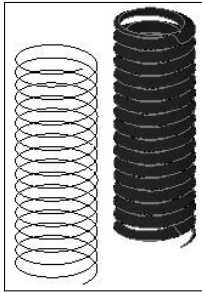
## Helix

The helix object in AutoCAD was first introduced with AutoCAD 2007, and it allows you to create a wireframe object that looks much like a spring or spiral. A helix isn't a



commonly used object by itself, but it is used as a path when sweeping or lofting a profile to create a 3D model of a spring, coil, or the threads of a fastener like a bolt or screw. For more information on sweeping or lofting profiles, see Chapter 8. Figure 6-4 shows helix objects that are paths to create a 3D representation of threads for a bolt.

**Figure 6-4:**  
3D threads  
created  
using  
helixes as  
the paths for  
sweeping  
and lofting  
techniques.

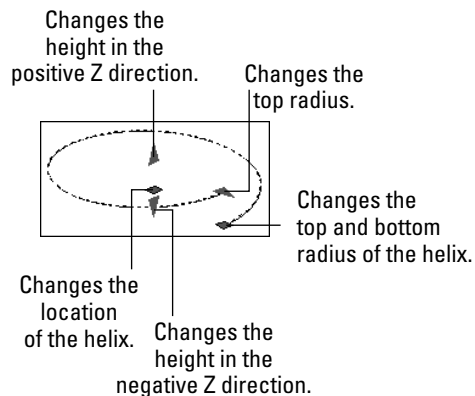


Helixes are defined with the following characteristics:

- ✓ **Turns:** Overall number of turns of the spiral that makes up the helix.
- ✓ **Turn Height:** Height difference between two turns of the helix.
- ✓ **Height:** Overall height of the helix in the Z direction from the start to end point.
- ✓ **Top Radius:** The radius of the top turn of the helix.
- ✓ **Bottom Radius:** The radius of the bottom turn of the helix.
- ✓ **Twist:** Specifies the direction (clockwise or counterclockwise) in which the helix is turned in as it twisted.

You create a helix by choosing Draw⇨Helix or entering **helix** at the command prompt. You can modify helixes by using grip editing or the Properties palette. To edit a helix by using grips, select the helix with no active command and select one of the square grips to edit the radius of the helix or its center point. If you select one of the arrow grips, you can modify the top radius or the height of the helix in the positive or negative Z direction. Figure 6-5 shows the different grips and what they're used for.

**Figure 6-5:**  
Getting  
a grip on  
modifying  
a helix.





### Exercise 6-5: Creating and Modifying a Helix

For this exercise, open the drawing file `Completed6-3.dwg` or `Exercise6-5.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Then practice creating and modifying a helix.

1. Choose `Draw` → `Helix`.
2. At the `Specify center point of base:` prompt, enter **0,0,0** and press Enter.
3. At the `Specify base radius or [Diameter] <1.0000>:` prompt, enter **5.25** and press Enter.
4. At the `Specify top radius or [Diameter] <6.2500>:` prompt, press Enter.
5. At the `Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <1.0000>:` prompt, enter **H** and press Enter.
6. At the `Specify distance between turns <0.2500>:` prompt, enter **2.15** and press Enter.
7. At the `Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <1.0000>:` prompt, press Enter.
8. Select the helix.
9. Select the square grip that changes the radius of the helix, and drag away from the helix's center point. Make sure to turn on ortho mode.  
Click `ORTHO` on the status bar so the button is pressed in.
10. Enter **1** and press Enter to adjust the radius of the helix to 6.25.
11. Right-click and choose `Properties`.
12. In the `Properties` palette, click in the `Constrain` field and select `Turn Height`.
13. Click in the `Height` field and enter **36.55**.  
Notice that the number of turns is automatically updated to a value of 17. This is because the `Constrain` for the helix was set to the turn height that the helix is assigned.
14. Position the cursor over the drawing and press `Esc` to deselect the selected helix.
15. Save the drawing with the name `Completed6-5.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Making 3D objects out of 2D objects

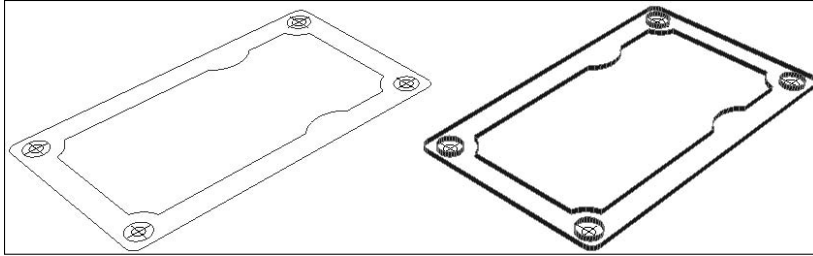
When you think of 3D modeling, the first things that might come to mind are faces and solids, but don't forget about 2D objects. You can use 2D objects to create 3D wireframes and even 3D surfaces and solids. If a line is extruded or is given a thickness, it's like a face of a surface model, whereas if you give a circle thickness, it's displayed like a hollow tube with no wall thickness; if extruded, you create a 3D solid because it's a closed object. Based on how you created your 2D drawings, you could start creating a 3D model based on your 2D design by extruding and giving objects thickness.

### Thickness

*Thickness* is a property that you can assign to many basic 2D objects, such as circles, lines, polylines, and ellipses, to give the objects height. Although an object that is given thickness is still a 2D object, it does appear like an object created as a surface model that

is left open on the top and bottom (see Figure 6-6). By default, all 2D objects are drawn with a thickness of 0 (zero), but you can set the default thickness value by using the THICKNESS system variable. To set the default thickness value, enter **thickness** at the command prompt and then enter the new value. Commonly, thickness is assigned to an object after it has been created using the Properties palette.

**Figure 6-6:**  
2D objects  
before and  
after a  
thickness is  
applied.



3. Which two commands convert 2D objects with thickness to a 3D solid or surface?
  - A. CONVTSOLID and CONVTSURFACE.
  - B. CHANGETOSOLID and CHANGETOSURFACE.
  - C. There are no commands that can be used to convert 2D objects with thickness to 3D solids or surfaces.

### Exercise 6-6: Giving 2D Objects Thickness

For this exercise, open the drawing file `Exercise6-6.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Then practice applying a thickness to 2D objects.



1. Select all the objects in the drawing using either an implied crossing window or a window selection method or press `Ctrl+A` to select all the objects in the drawing.
 

With no command active, select a point above and to the left of the objects in the drawing and then a point below and to the right of the objects.
2. Right-click and choose Properties.
3. In the Properties palette, click in the Thickness field and enter **1.7**.
 

Notice that the objects now come up off the plane into the Z direction.
4. Position the cursor over the drawing and press `Esc` to deselect the selected objects.
5. Close the Properties palette.
6. At the command prompt, enter **thickness** and press Enter.
7. At the Enter new value for THICKNESS <0.0000>: prompt, enter **1.7** and press Enter.
8. Choose `Draw` → `Circle` → `Center, Radius`.
9. At the Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: prompt, hold down the Shift key and right-click.
10. From the shortcut menu, choose Node.

11. At the `_nod` of prompt, select the point object that doesn't have a circle around it.
12. At the `Specify radius of circle or [Diameter] <0.0000>:` prompt, enter **3.75** and press Enter.  
The circle that you created was assigned a thickness value of 1.7 because you set the default thickness value for new objects that are created when you specified a value for the THICKNESS system variable.
13. Save the drawing with the name `Completed6-6.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

### ***Extrude***

*Extruding* an object is similar in concept to adding thickness to an object with several differences. When you add thickness to an object, it remains a circle or a line no matter what the thickness value is. If you extrude a 2D object, AutoCAD creates a new 3D object that is either a surface or a 3D solid based on whether the original 2D object was open or closed. If an open 2D object is extruded, a surface is created; if a surface is closed, a 3D solid is created instead. You can extrude these objects:

Lines	2D and 3D splines
Circles	Planar 3D faces
Arcs	Traces
Ellipses	Regions
Elliptical arcs	Planar surfaces
2D and 3D polylines	Planar faces on a 3D solid

When extruding a 2D object, you can specify a height or distance for the extrusion, a path, and a taper angle. A path for an extruded object can be one of the following common objects:

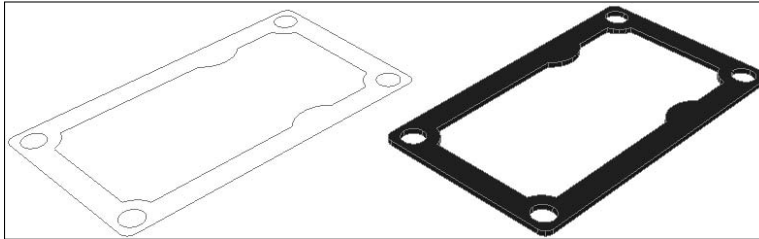
Lines	2D and 3D polylines
Circles	2D and 3D splines
Arcs	Edge of a solid
Ellipses	Edge of a surface
Elliptical arcs	Helix

To create an extruded object, you can do any of the following:

- ✓ Choose `Draw` ⇨ `Modeling` ⇨ `Extrude`.
- ✓ Click EXTRUDE on the Modeling toolbar.
- ✓ Click EXTRUDE on the 3D Make panel of the Dashboard.
- ✓ Enter **extrude** at the command prompt.

If you select a 2D object to extrude, the object is deleted from the drawing. If you don't want to delete the 2D object from the drawing, you have to set the Deletion Control While Creating 3D Objects option in the Options dialog box on the 3D Modeling tab to Retain Defining Geometry, or you must set the DELOBJ system variable to a value of 0 (zero). Figure 6-7 shows a closed polyline (rectangle) extruded using a 3D polyline as a path to create a bent bar or rail.

**Figure 6-7:**  
A region  
object  
before and  
after being  
extruded.



### Exercise 6-7: Extruding 2D Objects

For this exercise, open the drawing file `Exercise6-7.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents to practice extruding a 2D object.

1. Choose Draw↔Modeling↔Extrude.
2. At the `Select objects to extrude:` prompt, select the 2D object in the drawing that represents a gasket and press Enter.
3. At the `Specify height of extrusion or [Direction/Path/Taper angle] <0.0000>:` prompt, enter **1.7** and press Enter.

The 2D object is extruded 1.7 units in the positive Z-direction, giving it a consistent thickness.

4. Save the drawing with the name `Completed6-7.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Modeling with Regions

*Regions* are 2D versions of 3D solids. Regions are flattened 3D solids that can hide objects beyond them, and you can add and subtract them from other regions to create complex 2D objects. You can also create a region based on the intersection of two regions. Regions are created from closed shapes made from 2D objects — or loops. A *loop* is a series of objects that form a closed object. The closed object doesn't necessarily have to be a closed object itself like a circle, an ellipse, or a closed polyline. You can create regions from the following objects when they form a closed loop:

Lines	Elliptical arcs
Circles	2D polylines
Arcs	2D splines
Ellipses	Traces

### Creating regions

You create regions by using two different techniques: the REGION command and the Region option of the BOUNDARY command:

- ✓ The REGION command allows you to create region objects based on either the closed objects that you select (such as circles or closed 2D polylines) or 2D objects that form a closed boundary (such as lines that met endpoint to endpoint to create a closed area). Each of the closed objects and boundaries that the

REGION command extracts is known as a *loop*. For each loop that is extracted, a region object is created based on the shape and size of the closed object or boundary. You can start the REGION command by choosing Draw→Region or entering **region** at the command prompt.

- ✓ The BOUNDARY command works similarly to the way you pick points to determine the hatch boundary when using the HATCH command, but instead a region is created based on the calculated boundary. The BOUNDARY command (choose Draw→Boundary) displays the Boundary Creation dialog box (see Figure 6-8), where you can create regions based on closed objects.

**Figure 6-8:**  
The  
Boundary  
Creation  
dialog box  
allows you  
to create  
regions  
based on  
loops.



### Exercise 6-8: Creating a Region



For this exercise, open the drawing file `Exercise6-8.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Then practice creating regions from closed 2D objects.

1. Choose Draw→Region.
2. At the `Select objects:` prompt, pick a point above and to the left of the 2D objects to create window selection.
3. At the `Specify opposite corner:` prompt, pick a point below and to the right of the 2D objects to complete the window selection.
4. Press Enter to end selecting objects.

The following messages are displayed at the command prompt:

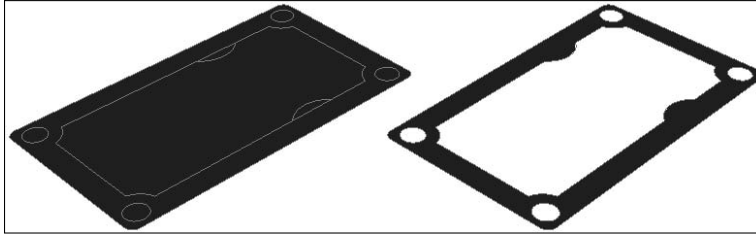
```
9 loops extracted.
9 Regions created.
```

5. Save the drawing with the name `Completed6-8.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Modifying regions

You can modify regions just as you do 2D objects in a drawing using basic modify operations, such as move, rotate, and copy. However, you can use a few other modify operations with regions: union, subtract, and intersect. (The union, subtract, and intersect operations work on 3D solids as well.) Figure 6-9 shows a number of regions before using the UNION and SUBTRACT commands to combine and remove regions from each other.

**Figure 6-9:**  
A number  
of regions  
before and  
after using  
UNION and  
SUBTRACT.



### *Union*

The UNION command allows you to combine more than one region to create a single region. By combining multiple regions, you create a complex 2D object that you can later use to create a 3D solid by extruding, lofting, or sweeping it. Examples of when you might want to create complex 2D regions are for gaskets or mounting brackets to which you can easily add holes or other features, or even for profiles that are used in revolving operations to create a 3D model of a part. You can start the UNION command by choosing **Modify**→**Solid Editing**→**Union** or by entering **union** at the command prompt.

### *Subtract*

To remove parts of a region, you don't use the TRIM command, but instead you use the SUBTRACT command. The SUBTRACT command allows you to remove one or more regions that intersect or overlap a region. By removing parts of a region, you can create holes or notches in a 2D region before extruding, lofting, or sweeping it to create a 3D object. You can start the SUBTRACT command by selecting the SUBTRACT command from the Modeling toolbar or entering **subtract** at the command prompt.

### *Intersect*

The INTERSECT command allows you to create a region based on where two or more regions overlap each other. Using INTERSECT is not very common when working with regions but is when working with 3D solids to subtract out a part of a solid that interferes with another solid. You can start the INTERSECT command by selecting INTERSECT on the 3D Make panel of the Dashboard or by entering **intersect** at the command prompt.

---

## **Exercise 6-9: Modifying Regions**



For this exercise, open the drawing file `Completed6-8.dwg` or `Exercise6-9.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Then practice unioning and subtracting regions.

1. Freeze the 3DConst layer.



- Display the Layers toolbar if it isn't currently displayed by right-clicking a toolbar and choosing Layers from the shortcut menu. Click the sun icon next to the layer 3DConst from the Layers drop-down list. The icon changes to a snowflake, indicating that the layer is now frozen.
2. Choose **Modify**→**Solid Editing**→**Subtract**.
3. At the **Select solids and regions to subtract from...** **Select objects:** prompt, select the outermost object and press Enter.

4. At the `Select solids and regions to subtract...` prompt, select the large innermost object and each of the four circles.
5. Press Enter to finish selecting objects.
6. Select the innermost object and choose `Modify↔Erase`.
7. Choose `Modify↔Solid Editing↔Union`.
8. At the `Select objects:` prompt, select the three regions and press Enter.
9. Choose `Modify↔Erase`.
10. At the `Select objects:` prompt, select the complex region and press Enter.  
Notice that the entire object is deleted because all the separate regions have been removed due to subtracting or unioning the regions with the main region.
11. Choose `Edit↔Undo Erase`.  
The erased region is brought back.
12. Save the drawing with the name `Completed6-9.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.



4. Exercise 6-9 shows how to union and subtract a region. What menu command do you choose to intersect a region?

## Getting information from a region

As you might have noticed, regions are special objects based on how you've created and modified them. Regions allow you to collect special information about them because they are complex 2D objects that you can use to construct 3D solids. You use the `LIST` command and Properties palette to obtain the area, perimeter, and the *bounding box* of a region — the bounding box of a region is an invisible box that the region fits into.

The `MASSPROP` command allows you to get even more information about a region or 3D solid. You can start the `MASSPROP` command in the following ways:

- ✓ Choose `Tools↔Inquiry↔Region/Mass Properties`.
- ✓ Click the `Region/Mass Properties` command on the Inquiry toolbar.
- ✓ Enter **massprop** at the command prompt.

You collect the following information about a region with this command:

- ✓ **Centroid:** A 2D or 3D coordinate value that represents the center point of the area for a region or 3D solid.
- ✓ **Moments of inertia:** Determines how the distribution of the mass is done around the X, Y, and Z axes. Values are often used to calculate things like the distributed loads for the bending and twisting forces generated inside a beam or the pressure of fluid on a plate.
- ✓ **Product of inertia:** Determines the force that causes the motion of an object with respect to two planes that are perpendicular to each other (XY, YZ, or ZX). Values are often used to perform stress analysis.



- ✓ **Radii of gyration:** Another set of values that indicates the moments of inertia.
- ✓ **Principal moments and X-Y directions about centroid:** Determines the axis through the centroid in which the movements of inertia are at the highest and the axis in which they're at the lowest.

The following is an example of the information that is displayed when using the MASSPROP command:

```

----- REGIONS -----
Area:                2388.8453
Perimeter:           685.7656
Bounding box:        X: 0.0000  -- 106.0000
                    Y: 0.0000  --  61.0000
Centroid:            X: 53.5588
                    Y: 30.5000
Moments of inertia:  X: 3497832.2813
                    Y: 9935701.2822
Product of inertia:  XY: 3902281.6661
Radii of gyration:   X: 38.2653
                    Y: 64.4919
Principal moments and X-Y directions about centroid:
                    I: 1275608.8982 along [1.0000 0.0000]
                    J: 3083193.8712 along [0.0000 1.0000]

Write analysis to a file? [Yes/No] <N>:

```

Before you exit the MASSPROP command, you have the option of creating a file with the analysis information that is displayed to the Text Window. You're prompted for a file name and location to save a file with the MPR extension to. You can then open the file with a standard text editor like Notepad or a word processor like Microsoft Word.

### Exercise 6-10: Collecting Information About a Region

For this exercise, open the drawing file `Completed6-9.dwg` or `Exercise6-10.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Then practice collecting information about the region.

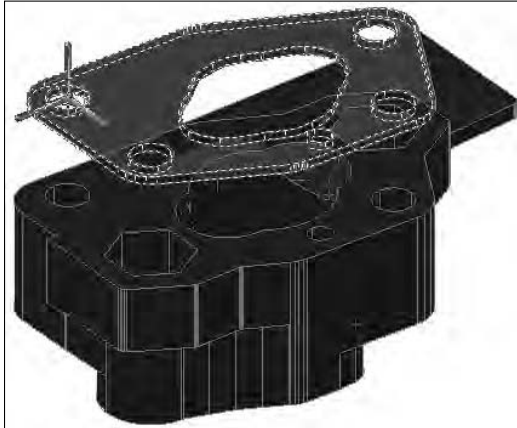
1. Choose Tools⇨Inquiry⇨Region/Mass Properties.
2. At the `Select objects:` prompt, select the region and press Enter.
3. At the `Write analysis to a file? [Yes/No] <N>:` prompt, enter N and press Enter.
4. Close the drawing without saving the changes.

## Modifying 3D Objects

You'll often find yourself spending more time modifying objects than creating them in your drawing. Although most of the modify commands that you have grown attached to in AutoCAD are available for use when working in 3D, some are specific to working in 3D. AutoCAD comes with 3D versions of the modify commands MOVE, ROTATE, MIRROR, and ARRAY. Two additional commands can help you move and rotate objects in one operation: ALIGN and 3DALIGN.

## 3D Move

You can move objects from one location to another by using a number of different methods, but the 3DMOVE command allows you to constrain objects to one axis when moving them. The 3DMOVE command uses what's known as a *gizmo* (see Figure 6-10). The gizmo for the 3DMOVE command is color coded to tell you which axis is going in which direction; when you pick a base point for the gizmo, you can then specify the axis you want the move operation to be constrained to.



**Figure 6-10:** Gizmo used to constrain objects when being moved in 3D.

You can start the 3DMOVE command in the following ways:

- ✓ Choose **Modify**→**3D Operations**→**3D Move**.
- ✓ Click **3D Move** on the Modeling toolbar or **3D Make** panel on the Dashboard.
- ✓ Enter **3dmove** at the command prompt.

Other ways move objects in a drawing are the following:

- ✓ The **Move** option with grip editing.
- ✓ The **ALIGN** or **3DALIGN** commands.
- ✓ Select objects and then drag them with the right mouse button. After you release the mouse button, select **Move Here**.

---

### Exercise 6-11: Moving an Object in 3D

For this exercise, open the drawing file `Exercise6-11.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Then practice moving objects using the move gizmo of the 3DMOVE command.



1. Choose **Modify**→**3D Operations**→**3D Move**.
2. If the **Selecting Subobjects on Solids** dialog box is displayed, click **Close**.
3. At the **Select objects:** prompt, select the gasket (object in red) above the main part and press **Enter**.

4. At the `Specify base point or [Displacement] <Displacement>`: prompt, hold down the Shift key and right-click.
5. From the shortcut menu, click Center.
6. At the `_cen` of prompt, select the center of the hole in the upper-left corner of the gasket.
7. Position the crosshairs over the Z axis indicator of the gizmo and click when it turns yellow.

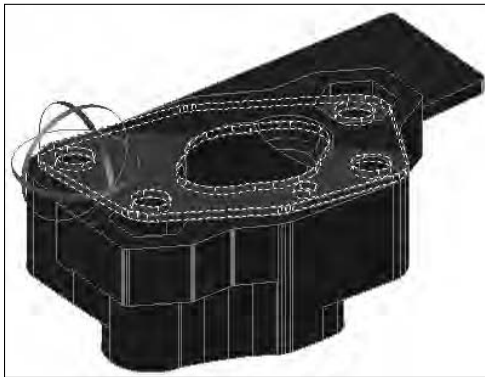
No matter where you drag in the drawing, the object is constrained to the Z axis.

8. At the `Specify second point or <use first point as displacement>`: prompt, drag the crosshairs down from the base point. Enter **12** and then press Enter.
9. Save the drawing with the name `Completed6-11.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

## 3D Rotate

You can rotate objects around a base point by using a number of different methods, but the 3DROTATE command allows you to constrain the rotation of objects to one specific axis. The 3DROTATE command uses a gizmo (see Figure 6-11) like the 3DMOVE command does. The gizmo for the 3DROTATE command is color coded to tell you the direction in which an axis is going; when you pick a base point for the gizmo, you can then specify the axis you want the rotate operation to be constrained to.

**Figure 6-11:**  
Gizmo used  
to constrain  
objects  
when being  
rotated  
in 3D.



You can start the 3DROTATE command in the following ways:

- ✓ Choose `Modify` → `3D Operations` → `3D Rotate`.
- ✓ Click 3D Rotate on the Modeling toolbar or 3D Make panel on the Dashboard.
- ✓ Enter **3drotate** at the command prompt.



The ROTATE3D command is the older version of the 3DROTATE command, but it does offer a few different options from the 3DROTATE command. One of the different options that the ROTATE3D command offers is the ability to define the axis for rotating the selected objects by an object, the last object, the current view, and by a specific axis.

You can rotate objects in a drawing in the following ways:

- ✓ Enter **rotate** at the command prompt to rotate objects along the current XY plane.
- ✓ Enter **3drotate** at the command prompt to rotate objects in 3D space.
- ✓ Use the Rotate option when using grip editing.
- ✓ Enter **align** or **3dalign** at the command prompt to rotate objects in either 2D or 3D space.



### Exercise 6-12: Rotating an Object in 3D

For this exercise, open the drawing file `Exercise6-12.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Then practice rotating objects using the rotate gizmo of the 3DROTATE command.

1. Choose `Modify` → `3D Operations` → `3D Rotate`.
2. If the `Selecting Subobjects on Solids` dialog box is displayed, click `Close`.
3. At the `Select objects:` prompt, select the gasket (object in red) above the main part and press `Enter`.
4. At the `Specify base point:` prompt, hold down the `Shift` key and right-click.
5. From the shortcut menu, select `Center`.  
The `Center` object snap allows you to override the current running object snaps with the `Center` object snap only.
6. At the `_cen` of prompt, select the center at the bottom of the hole in the upper-left corner of the gasket.
7. At the `Pick a rotation axis:` prompt, position the crosshairs over the green ribbon that runs around the Y axis of the gizmo and click when it turns yellow.
8. At the `Specify angle start point or type an angle:` prompt, enter `-5` and press `Enter`.
9. Repeat the 3DROTATE command and select the gasket again. Use the same point for the base point as the previous rotation operation.
10. At the `Pick a rotation axis:` prompt, position the crosshairs over the ribbon that runs around the X axis of the gizmo and click when it turns yellow.
11. At the `Specify angle start point or type an angle:` prompt, enter `-5` and press `Enter`.
12. Save the drawing with the name `Completed6-12.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

## 3D Align and Align

When working in 3D, you often find the need to align fasteners to a hole in a part or where two parts might need to match up. AutoCAD offers two different methods to align objects. Both the 3DALIGN and ALIGN commands allow you to move and rotate objects based on source and destination points:

- ✓ **3DALIGN:** Choose `Modify` → `3D Operations` → `3D Align` or enter **3dalign** at the command prompt. You're prompted to specify up to three base points and then up to three destination points. The command also allows you to create a copy of the selected objects before starting the alignment process.

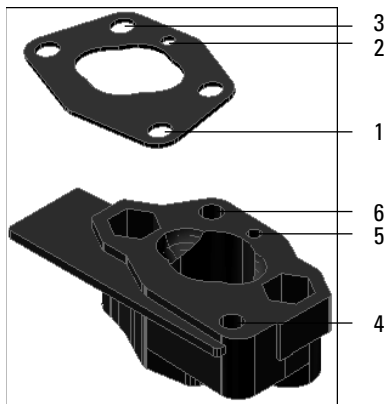
- ✓ **ALIGN:** Choose **Modify**→**3D Operations**→**Align** or enter **align** at the command prompt. You're prompted for up to three pairs of alignment points. Unlike with the **3DALIGN** command, you specify a source point and then the matching destination point before specifying the next source point.



### Exercise 6-13: Aligning Objects in 3D with the 3DALIGN Command

For this exercise, open the drawing file `Exercise6-13.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Then practice rotating and moving objects into place with the **3DALIGN** command.

1. Choose **Modify**→**3D Operations**→**3D Align**.
2. At the **Select objects:** prompt, select the gasket (object in red) above the main part and press **Enter**.
3. At the **Specify base point or [Copy]:** prompt, hold down the **Shift** key and right-click.
4. From the shortcut menu, click **Center**.
5. At the **\_cen** of prompt, use the **Center** object snap and select the bottom of the hole indicated by 1 in Figure 6-12.



**Figure 6-12:**  
Aligning  
two parts  
based on  
the holes in  
both parts.

6. At the **Specify second point or [Continue] <C>:** prompt, use the **Center** object snap and select the bottom of the hole indicated by 2 in Figure 6-12.
7. At the **Specify third point or [Continue] <C>:** prompt, use the **Center** object snap and select the bottom of the hole indicated by 3 in Figure 6-12.
8. At the **Specify first destination point:** prompt, use the **Center** object snap and select the top of the hole indicated by 4 in Figure 6-12.
9. At the **Specify second destination point or [eXit] <X>:** prompt, use the **Center** object snap and select the top of the hole indicated by 5 in Figure 6-12.
10. At the **Specify third destination point or [eXit] <X>:** prompt, use the **Center** object snap and select the top of the hole indicated by 6 in Figure 6-12.
11. Save the drawing with the name `Completed6-13.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

---

**Exercise 6-14: Aligning Objects in 3D with the ALIGN command**

Open the drawing file `Exercise6-13.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Use the ALIGN command to align the gasket with the main part instead of the 3DALIGN command. Save the completed drawing as `Completed6-14.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents.

## Mirror 3D

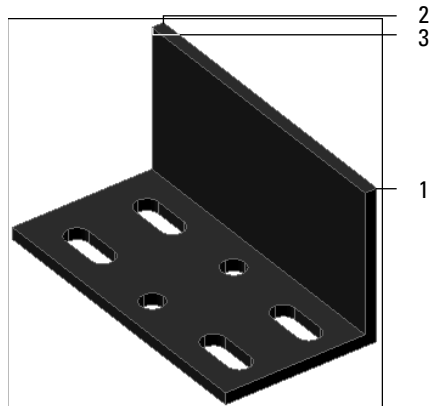
Mirroring objects allows you to build 3D models with symmetry. Based on how your model is designed, you might be able to mirror objects along a plane to create the other half of the object. It's best to use the MIRROR command when drafting in 2D because it's limited to the XY plane. The MIRROR3D command, however, allows you to mirror along a specific plane or axis that might not be on the XY plane. With the MIRROR3D command, you can use an object, the last object in the drawing, the current view, or three points to specify the plane to mirror the selected objects. You can also use the Z axis or the XY, YZ, or ZX plane to mirror the selected objects. You start the MIRROR3D command by choosing `Modify` → `3D Operations` → `3D Mirror` or entering `mirror3d` at the command prompt.

---

**Exercise 6-15: Mirroring an Object in 3D**

For this exercise, open the drawing file `Exercise6-15.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Then practice mirroring objects in 3D using the MIRROR3D command to create a symmetrical part.

1. Choose `Modify` → `3D Operations` → `3D Mirror`.
2. At the `Select objects:` prompt, select the bracket and press Enter.
3. At the `Specify first point of mirror plane (3 points) or [Object/Last/Zaxis/View/XY/YZ/ZX/3points] <3points>:` prompt, hold down the Shift key and right-click.
4. From the shortcut menu, choose `Endpoint` and select the corner indicated by 1 in Figure 6-13.



**Figure 6-13:**  
Using the MIRROR3D command to mirror half of a bracket to create the final model.

5. At the `Specify second point of mirror plane:` prompt, use the Endpoint object snap and select the corner indicated by 2 in Figure 6-13.
6. At the `Specify third point of mirror plane:` prompt, use the Endpoint object snap and select the corner indicated by 3 in Figure 6-13.
7. At the `Delete source objects? [Yes/No] <N>:` prompt, press Enter.
8. Use the UNION command to join the two solids together.  
Choose `Modify`→`Solid Editing`→`Union` and then select both halves of the part.
9. Save the drawing with the name `Completed6-15.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

## 3D Array

Mirroring objects makes it easy to create symmetrical objects, whereas arraying objects allows you to create repeating patterns. The ARRAY command allows you to array objects in a rectangular or polar pattern on the XY plane, but the 3DARRAY command allows you to create rectangular arrays with an additional direction, and the polar option allows you to specify the axis of rotation for the array. When creating a rectangular array with the 3DARRAY command when compared to the ARRAY command, you have an additional direction for the array known as a *level* — which is the number of times you want the array to be defined in the Z direction. You start the 3DARRAY command by choosing `Modify`→`3D Operations`→`3D Array` or entering `3darray` at the command prompt.

---

### Exercise 6-16: Creating a 3D Rectangular Array

For this exercise, open the drawing file `Exercise6-16.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D array of holes in a rail.

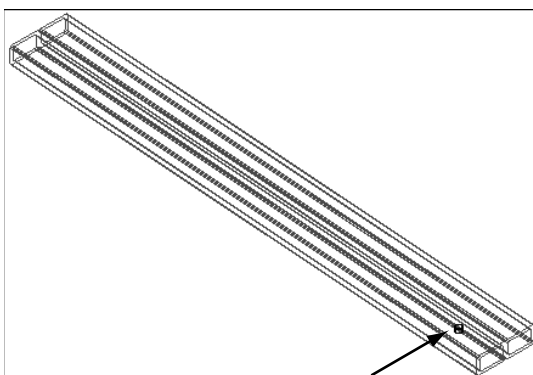
1. Choose `Modify`→`3D Operations`→`3D Array`.
2. At the `Select objects:` prompt, select the cylinder in the lower-left corner of the rail and press Enter.

The cylinder is indicated by the arrow in Figure 6-14.

---

**Figure 6-14:**  
Adding mounting holes to a rail with the rectangular option of the 3DARRAY command.

---



3. At the Enter the type of array [Rectangular/Polar] <R>: prompt, enter **R** and press Enter.
4. At the Enter the number of rows (---) <1>: prompt, enter **11** and press Enter.
5. At the Enter the number of columns (|||) <1>: prompt, enter **2** and press Enter.
6. At the Enter the number of levels (...) <1>: prompt, enter **2** and press Enter.
7. At the Specify distance between rows (---): prompt, enter **1.5** and press Enter.
8. At the Specify distance between columns (|||): prompt, enter **1.03** and press Enter.
9. At the Specify distance between levels (...): prompt, enter **-0.555** and press Enter.
10. Subtract the cylinders that you just arrayed from the rail.  
Choose Modify→Solid Editing→Subtract. Select the rail to subtract the cylinders from and then select the cylinders when prompted for the objects to subtract.
11. Apply the Realistic visual style so you can see that there are now holes in the rail.  
Choose View→Visual Styles→Realistic.
12. Save the drawing with the name `Completed6-16.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

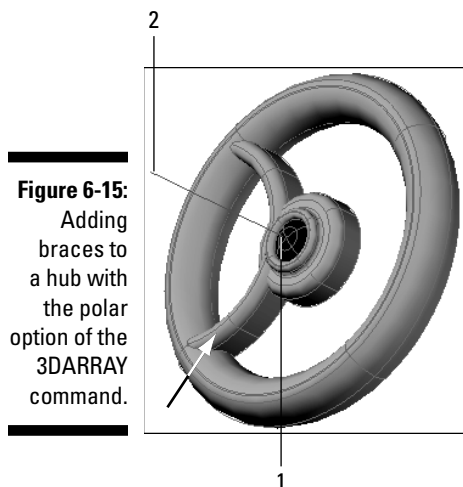
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### Exercise 6-17: Creating a 3D Polar Array

For this exercise, open the drawing file `Exercise6-17.dwg` from the Chapter 6 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D array of braces around a specific axis to complete the hub.

1. Choose Modify→3D Operations→3D Array.
2. At the Select objects: prompt, select the brace on the inside of the wheel and press Enter.

The brace is indicated by the arrow in Figure 6-15.





3. At the `Enter the type of array [Rectangular/Polar] <R>:` prompt, enter **P** and press Enter.
4. At the `Enter the number of items in the array:` prompt, enter **3** and press Enter.
5. At the `Specify the angle to fill (+=ccw, -=cw) <360>:` prompt, enter **360** and press Enter.
6. At the `Rotate arrayed objects? [Yes/No] <Y>:` prompt, enter **Y** and press Enter.
7. At the `Specify center point of array:` prompt, use the Endpoint object snap and select the end of the line closest to the center of the hub.  
The end of the line is indicated by 1 in Figure 6-14.
8. At the `Specify second point on axis of rotation:` prompt, use the Endpoint object snap and select the end of the line farthest away from the center of the hub.  
The end of the line is indicated by 2 in Figure 6-14.
9. Use the UNION command to join the two new braces and the original brace to the inner and outer parts of the wheel.  
Choose `Modify`⇨`Solid Editing`⇨`Union` and then select the five solids to union together.
10. Save the drawing with the name `Completed6-17.dwg` to the Chapter 6 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Answers to Practice Opportunities

1. To find which layer is current, look at the current value displayed on the Layers drop-down list on the Layers toolbar or enter status at the command prompt.
2. Why would you want to use a 3D polyline?  
They're often used as paths for lofting or extruding objects and creating 3D wireframe models.
3. Which two commands convert 2D objects with thickness to a 3D solid or surface?  
A. You can use the commands CONVTSOLID and CONVTSURFACE to convert some open and closed 2D objects with thickness to a solid or surface. CONVTSOLID converts a closed polyline or circle to a solid. CONVTSURFACE converts a line, an arc, or an open zero-width polyline to a surface.
4. What menu command do you choose to intersect a region?  
Choose `Modify`⇨`Solid Editing`⇨`Intersect`. At the `Select objects` prompt, select the objects you want to join together.



## Chapter 7

# Additional Exercises: Making the Transition

---

### *In This Chapter*

- ▶ Reinforcing concepts used to navigate 3D models
  - ▶ Reinforcing concepts used in moving from 2D to 3D
- 

**I**n the previous chapters of Part II, you practice some of the basic concepts of working in the third dimension, viewing a drawing, and performing some basic creation and modification of 3D objects. The exercises in this chapter have you working with a small rear assembly of a custom motorcycle. By using a small assembly of parts, you get comfortable working with many different parts in a drawing at a time and establish a good understanding of the workflow when working with more than one part at a time.

## *Navigating a 3D Model*

In this series of exercises, you work with the assembly of parts that make up the rear assembly of a custom motorcycle. You navigate and use some of the different visualization features from preceding chapters in this part of the book.

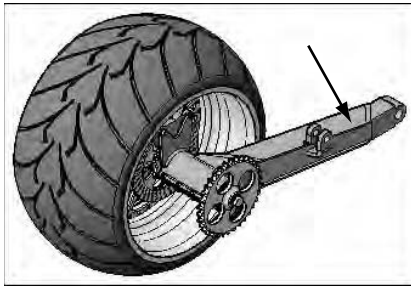
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### **Exercise 7-1: Navigating an Assembly of Parts**

For this exercise, open the drawing file `Exercise7-1.dwg` from the Chapter 7 folder in the My Practice Drawings folder under My Documents. Then practice navigating around the assembly.

1. View the assembly from the left by choosing `View↔3D View↔Left`.  
A number of parts show through the openings on the rim.
2. Select the swing arm (indicated by the arrow in Figure 7-1) and choose `View↔Orbit↔Constrained Orbit`.  
All the parts except for the swing arm are hidden.
3. Click and drag in the drawing window in various different directions to view the swing arm from different angles.
4. With the constrained orbit still active, right-click and choose `Preset Views↔SE Isometric`.
5. Press `Esc` to exit the constrained orbit mode.  
All the other parts of the assembly return.

**Figure 7-1:**  
The rear  
assembly of  
a custom  
motorcycle.



Model courtesy of Mark Hechel of MasterGraphics, Inc.

6. Change the visual style of the current view to Realistic.
7. Save the drawing with the name `Completed7-1.dwg` to the Chapter 7 folder in the My Practice Drawings folder under My Documents and close the drawing.

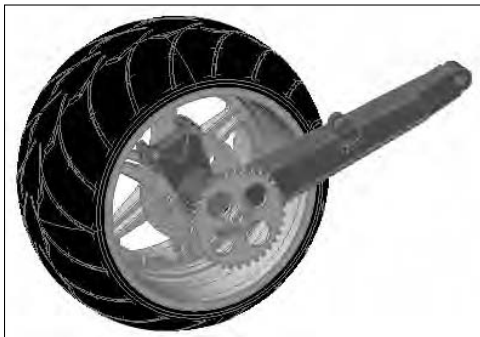


### Exercise 7-2: Creating a Named View with a Gradient Background

For this exercise, open the drawing file `Exercise7-2.dwg` from the Chapter 7 folder in the My Practice Drawings folder under My Documents. Then practice creating a named view and assign a background with a two-color gradient fill to the named view.

1. Use some of the 3D navigation commands to adjust the view of the model so you can see the parts that are between the rim and the swing arm. Try to adjust the view so it looks somewhat similar to the one of the assembly in Figure 7-2.

**Figure 7-2:**  
Getting  
up close  
with the  
assembly.



Model courtesy of Mark Hechel of MasterGraphics, Inc.

2. Choose **View** ⇨ **Named Views** to bring up the View Manager dialog box.
3. Click **New** and enter **Inside look** in the View Name text box of the New View dialog box.
4. Under the Settings section, deselect the Save Layer Snapshot with View check box and select Realistic from the Visual Style drop-down list.
5. Under the Background section, select Gradient from the drop-down list.
6. In the Background dialog box, deselect the Three Color check box under the Gradient Options section.
7. Click the Top Color swatch to display the Select Color dialog box.
8. In the Select Color dialog box, click the Index Color tab and select color 160, which is a blue color. Click OK to return to the Background dialog box.

9. Set the Bottom Color swatch to the color 250.
10. Click OK to return to the New View dialog box.
11. Click OK to return to the View Manager dialog box.
12. Click Set Current and then OK to exit the View Manager dialog box.

The assigned background is displayed in the current viewport.

13. Save the drawing with the name `Completed7-2.dwg` to the Chapter 7 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Taking a 2D Drawing and Creating a 3D Part

In this series of exercises, you create a new part from a 2D drawing and then place it in the drawing that contains the assembly. To create the new 3D part, you create a couple of regions and then extrude them to create the new part. You then reference the new part into the assembly and align it into the correct location where it's attached to the swing arm.

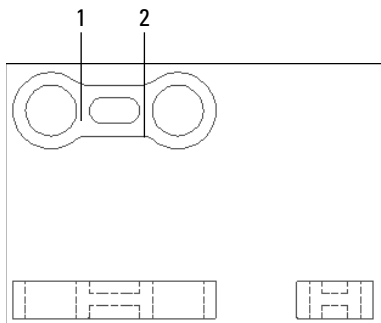


### Exercise 7-3: Starting with a 2D Drawing and Ending with a 3D Part

For this exercise, open the drawing file `Exercise7-3.dwg` from the Chapter 7 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D part from an existing 2D drawing.

1. Activate Model space by clicking the Model tab at the bottom of the drawing window or clicking the Model button on the status bar when the layout tabs aren't displayed.
2. Turn off the Dims layer and create two new layers. Name one of the new layers `Transfer_link` and the other `3Dconst`. Set the `Transfer_link` layer to color 45 and `3Dconst` to the color Red (or 1). When the new layers are set up, set the `Transfer_link` layer current.
3. Choose `Draw` → `Boundary`.
4. In the Boundary Creation dialog box, select `Region` from the Object Type drop-down list and click `Pick Points`.
5. Pick the point indicated by 1 in Figure 7-3 and press Enter.

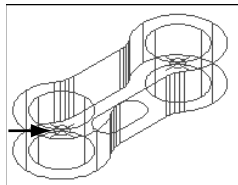
**Figure 7-3:**  
Creating a region from a closed boundary and extruding it to create a 3D part.



Model courtesy of Mark Hechel of MasterGraphics, Inc.

6. Choose **Modify**→**Solid Editing**→**Subtract**.
7. Select the outermost edge (indicated by 2) and press **Enter**.
8. Select the two large circles and press **Enter**.
9. Choose **Draw**→**Modeling**→**Extrude** and select the outermost edge (indicated by 2) of the large region. Press **Enter** to end object selection.
10. When prompted for the height of the extrusion, enter **0.75** and press **Enter**.  
You can turn on the Dims layer to verify what the height of the part should be if you want to.
11. Change to the SW Isometric view so you can see the new 3D part.  
Choose **View**→**3D View**→**SW Isometric**.
12. Turn off all the other layers in the drawing except for the layers **Transfer\_link** and **3Dconst**. Set the layer **3Dconst** current.
13. Set **Center** as the only running object snap and turn on object snap.
14. Choose **Draw**→**Point**→**Single** and hold down **Shift** while right-clicking. From the right-click menu, choose **Mid Between 2 Points**.
15. Pick the center point at the bottom of one of the holes when prompted for the first point and then the top of the same hole for the second point.
16. Repeat Steps 14 and 15 for the other hole in the 3D part.  
You now have two point objects in the center of the two holes, and your drawing should look like Figure 7-4.

**Figure 7-4:**  
The completed part.



*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

17. At the command prompt, enter **base**, and when prompted for the new base point, use the **Node** object snap and select the point object indicated by the arrow in Figure 7-5.
18. Save the drawing with the name **Completed7-3.dwg** to the Chapter 7 folder in the My Practice Drawings folder under My Documents and close the drawing.

### Exercise 7-4: Bringing the New Part and Assmebly Together



For this exercise, open the drawing file **Exercise7-4.dwg** or **Completed8-2.dwg** from the Chapter 7 folder in the My Practice Drawings folder under My Documents. Then practice bringing in an external drawing that contains a 3D part that needs to be added to the assembly. You can use either the **INSERT** (**Insert**→**Block**) command or the **XATTACH** (**Insert**→**DWG Reference**) command to merge the 3D part into the assembly. I show using the **XATTACH** command because it allows you to make changes to the part so the assembly can be updated at a later date if you need to.

1. Turn on the layer **3DConst**.
2. Choose **Insert**→**DWG Reference** to display the **Select Reference File** dialog box.

3. Browse to the Chapter 7 folder in the My Practice Drawings folder under My Documents and select `Transfer_link.dwg` (or `Completed7-3.dwg` if you did Exercise 7-3).
4. In the External Reference dialog box, set the following options:
  - Select Attachment under the Reference Type section.
  - Set Path Type to Relative Path.
  - Select Specify On-Screen under the Insertion Point section.
  - Deselect the other Specify On-Screen check boxes under the Scale and Rotation sections.
5. Click OK.
6. Pick a point near the swing arm.

The 3D part is now linked into the drawing, so any changes that you make in `Transfer_link.dwg` or `Completed7-3.dwg` are applied to the assembly. Your drawing should look similar to the one in Figure 7-5.



**Figure 7-5:**  
Part linked  
with the  
assembly.

*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

7. Save the drawing with the name `Completed7-4.dwg` to the Chapter 7 folder in the My Practice Drawings folder under My Documents and close the drawing.

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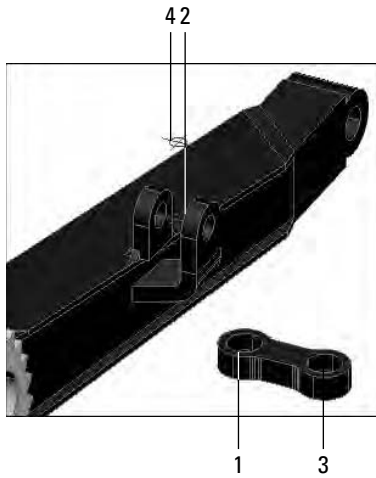
### Exercise 7-5: Aligning the Part



For this exercise, open the drawing file `Exercise7-5.dwg` from the Chapter 7 folder in the My Practice Drawings folder under My Documents. Then practice using the `ALIGN` command to move and rotate the part into position.

1. Set the Node as the running object snap and turn on object snap.
2. Choose `Modify` → `3D Operations` → `Align` and select the transfer link part that you attached in Exercise 7-4. Press `Enter` to end object selection.
3. When prompted for the first source point, pick the point object indicated by 1 in Figure 7-6.
4. Pick the point indicated by 2 for the first destination point.
5. Pick the point indicated by 3 for the second source point and then the point indicated by 4 for the second destination point.
6. When prompted for the third source point, press `Enter` twice to use the two points to align the transfer link.

**Figure 7-6:**  
Aligning the  
part into  
position  
on the  
assembly.



*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

7. Turn on the layers Transfer\_link\_bushings and Transfer\_link\_fasteners.

Mechanical fasteners and connectors for the transfer link were previously inserted into the drawing and are now displayed completing the connection (see Figure 7-7).

**Figure 7-7:**  
The com-  
pleted  
connection  
and align-  
ment of the  
transfer link.



*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

8. Save the drawing with the name Completed7-5.dwg to the Chapter 7 folder in the My Practice Drawings folder under My Documents and close the drawing.

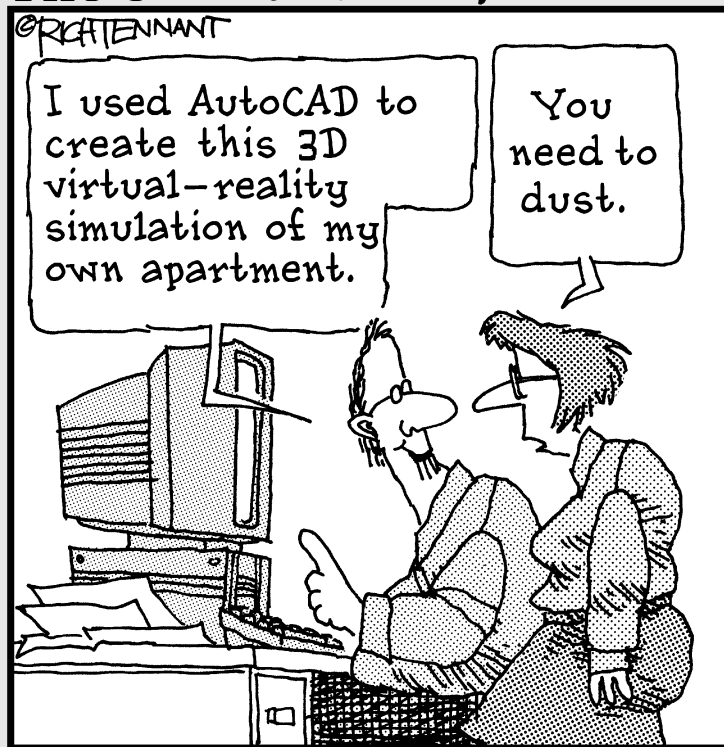


# Part III

## 3D Modeling — Solid Modeling

The 5<sup>th</sup> Wave

By Rich Tennant



## *In this part . . .*

**Y**ou can create solid models by using two different approaches. You can create 3D primitive geometry such as spheres and boxes, or you can use 2D and 3D objects with complex operations to create a 3D model. Most models that you create are based on a combination of both approaches. As you create 3D models with 3D solids, you find out how to create primitive and complex 3D solids and then modify them to add features such as holes, dimples, fillets and rounds, and bevels.

Although you might start with a 2D drawing to create a 3D model, AutoCAD allows you to create 2D objects from views of a 3D model created with 3D solids. The objects that are created from the views of a 3D model can be used to create a set of 2D drawings that contain hidden lines, hatches or fills, text, and dimensions to communicate how parts of a 3D model should be manufactured.

## Chapter 8

# Modeling with Solids

---

### *In This Chapter*

- ▶ Working with solids
  - ▶ Creating primitive solids
  - ▶ Creating complex solids
- 

**B**uilding 3D solids is the most flexible way to create 3D models in AutoCAD due to the ease of creating and modifying them. 3D solids were improved (more or less they saw a complete overhaul) in AutoCAD 2007 to be more flexible so they could be used to create both conceptual and dimensionally accurate models. The exercises in this chapter show you how to create primitive and complex 3D solids.

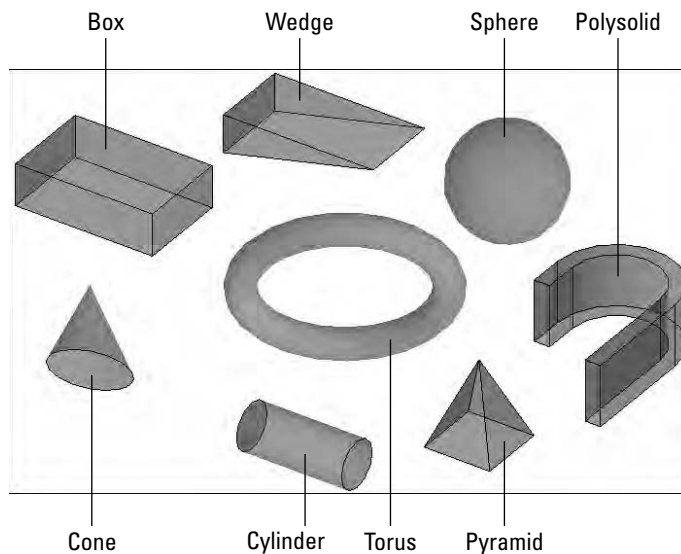
## *Working with 3D Solids*

In the preceding chapter, you create 3D objects by adding thickness to and extruding 2D objects to create a 3D solid. Although starting with 2D objects is one way to model in 3D, it isn't the only way. AutoCAD allows you to create two different classifications of 3D solids: primitive solids and complex solids. When you create a complex solid by extruding the region, it's a complex solid because it contains a number of features like holes and fillets.

### *Primitive solids*

*Primitive solids* (affectionately called *primitives*) are basic geometric shapes that create 3D models. AutoCAD allows you to create eight different 3D solid primitives (see Figure 8-1). The primitives fall into two categories: those with straight edges and those with curves. Okay, so most of the primitives fall into two categories nicely with the exception of what's called a polysolid. A *polysolid* is the 3D solid version of a polyline, which can be made up of both straight and curved segments.

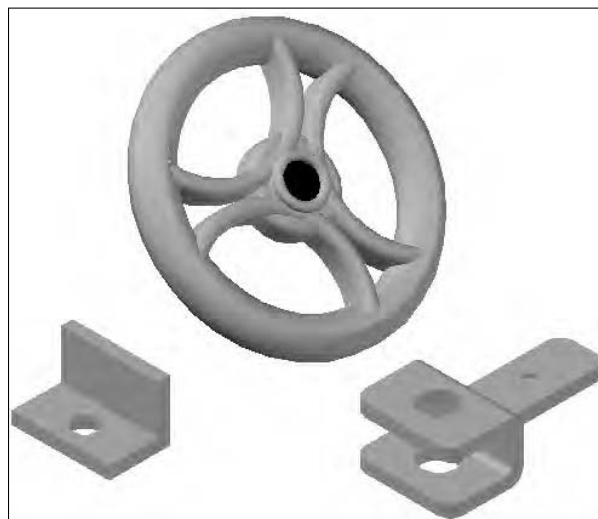
Prior to AutoCAD 2007, you could create 3D solid primitives. However, often the 3D solid suddenly got a case of amnesia and forgot what type of primitive it was. This problem made it very difficult to make changes to the 3D object. 3D solid primitives starting with AutoCAD 2007 could be modified using grips to change their size and shape, which gave users much greater control of doing conceptual models without having to resort to re-creating the object because it remembered what type of primitive it was.



**Figure 8-1:**  
3D solid  
primitives.

## Complex solids

*Complex solids* are created when you combine or subtract primitive solids from each other or create 3D solids from closed 2D objects (also known as profiles) through operations like lofting, sweeping, or extruding (see Figure 8-2). Chapter 6 covers adding two or more regions together to create a single region, subtracting one or more regions from a region, and creating a new region based on where two or more regions overlap; the same holds true for 3D solids. In Chapter 6, I introduce extruding a closed 2D object to create a new 3D solid. You can create complex solids later in this chapter, in the section “Nothing Complex about These Solids.” Most of the 3D models that you create will very likely be made up of complex solids that were created from primitives or that started off being 2D objects.



**Figure 8-2:**  
Complex  
3D solids  
created  
with various  
methods.

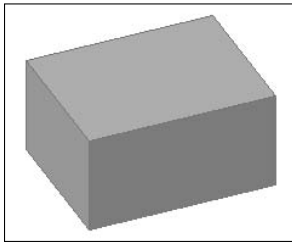
## Solids that Follow the Straight (And Narrow)

AutoCAD allows you to create four different types of primitives that have straight edges, and one of them can contain both straight and curved edges. The 3D solid primitives that have straight edges are box, wedge, pyramid, and polysolid.

### Box

A *box* is one of the more common types of primitive 3D solids that you can create. You can create a box as a cube with all sides being the same size or a rectangular shape with the ends being the same size (see Figure 8-3). When you have a box, you can use it to create a rail for a machine or create a mounting flange.

**Figure 8-3:** Boxes aren't only good for packing things; they're good for modeling, too.



You create a box in the following ways:

- ✓ Choose Draw→Modeling→Box.
- ✓ Click Box on the Modeling toolbar or the 3D Make panel on the Dashboard.
- ✓ Enter **box** at the command prompt.

### Exercise 8-1: Creating a 3D Solid Box

For this exercise, open the drawing file `Exercise8-1.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D solid primitive of a box.

1. Choose Draw→Modeling→Box.
2. At the `Specify first corner or [Center]:` prompt, enter **0,0,0** and press Enter to specify the first corner of the box.

You can also enter **C** to use the Center option, which allows you to create a box by specifying the length, width, and height of the box from its true center point.

3. At the `Specify other corner or [Cube/Length]:` prompt, drag the crosshairs to 155,200 and click to specify the width and length of the box.

You can also enter **C** for the Cube option to create a box with all equal sides or enter **L** to specify the length of the box.



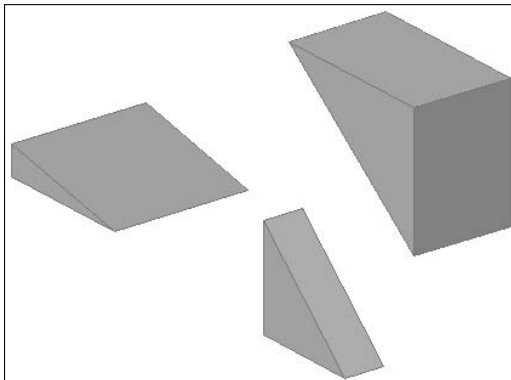
4. Enable Polar Tracking and set the snap mode to Polar.

To enable Polar Tracking, right-click the POLAR button on the status bar and choose On, click the POLAR button so it is down, or press F8. To enable Polar Snap, right-click the SNAP button on the status bar and choose Polar Snap On, or right-click the SNAP button and select Settings. In the Drafting Settings dialog box, select PolarSnap under the Snap Type section and click OK.

5. At the Specify height or [2Point] <5.0000>: prompt, drag the crosshairs up from the coordinate 155,200 and click when the tooltip reads Polar: 105.0000 < +Z to specify the height of the wedge.
6. Save the drawing with the name Completed8-1.dwg to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Wedge

Wedges are boxes that are cut in half diagonally from an upper to lower edge. When creating a wedge, you can create it as a cube with all sides being the same size, with the exception of the diagonal face or a rectangular shape (see Figure 8-4).



**Figure 8-4:**  
Variations of  
wedges that  
you can  
create.



1. When is a wedge useful?
  - A. For creating an incline or shim.
  - B. For creating the slot in a screwhead.
  - C. For creating curved profiles.

You can create a wedge with the following methods:

- ✓ Choose Draw ⇨ Modeling ⇨ Wedge.
- ✓ Click Wedge either on the Modeling toolbar or on the 3D Make panel on the Dashboard.
- ✓ Enter **wedge** at the command prompt.

---

### Exercise 8-2: Creating a 3D Solid Wedge

For this exercise, open the drawing file `Exercise8-2.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D solid primitive of a wedge.

1. Choose `Draw` → `Modeling` → `Wedge`.
2. At the `Specify first corner or [Center]:` prompt, enter `C` and press Enter to create a wedge from the center of the object.
3. At the `Specify corner:` prompt, drag the crosshairs to 35,35 and click to specify the width and length of the wedge.
4. At the `Specify other corner or [Cube/Length]:` prompt, drag the crosshairs to 70,70 and click to specify the length of the wedge.

You can also enter `C` for the Cube option to create a wedge with all equal sides or enter `L` to specify the length of the wedge.

5. At the `Specify height or [2Point] <5.0000>:` prompt, enter `15` and press Enter to specify the height of the wedge.
6. Use one of the orbiting commands to look at the wedge from different angles. Because you created the wedge by using the Center option, the object is created above and below the working plane.

To start one of the 3D orbiting commands, choose `View` → `Orbit` and then one of the items under the Orbit submenu.

Hold down the Shift key and middle mouse button to enter transparent orbiting.

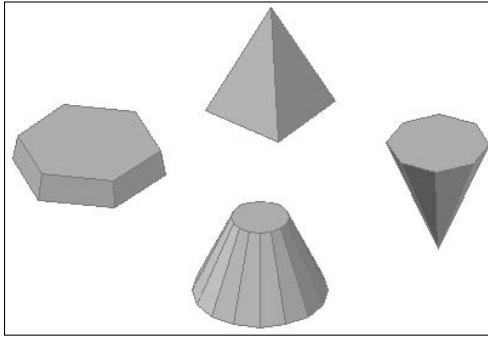
7. Save the drawing with the name `Completed8-2.dwg` to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.



## Pyramid

*Pyramids* are objects that have a flat base, a height, and can be made up of 3 to 32 sides that run up (or down) from the base to the top of the object. Pyramids can represent the head of a bolt or a custom fitting to the end of a crank or shaft. What makes a pyramid a unique and useful object is that it can have a point at the top (similar to a typical pyramid in Egypt) or it can be flat, which allows you to create a uniform polygon with multiple sides, forming a shaft that's hexagon shaped.

When creating a pyramid, you specify the center point of the base and then its radius value (which can be inscribed or circumscribed) like the process of creating a polygon with the `POLYGON` command. You can also define a pyramid by specifying the length of one of the edges of the base to calculate the overall base size for the pyramid. After you define the base of the pyramid, you're prompted to define its height and the top radius that defines whether it is created with a pointy or flat top. If you need a radius for the top, you're prompted to specify the height of the pyramid after the radius is specified. Figure 8-5 shows some of the different types of pyramids that you can create in AutoCAD.



**Figure 8-5:**  
Variations of  
pyramids  
that you can  
create.

To create a pyramid, start the command in any of the following ways:

- ✓ Choose Draw → Modeling → Pyramid.
- ✓ Click Pyramid either on the Modeling toolbar or on the 3D Make panel on the Dashboard.
- ✓ Enter **pyramid** at the command prompt.

### Exercise 8-3: Creating a 3D Solid Pyramid

For this exercise, open the drawing file `Exercise8-3.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D solid primitive of a pyramid.

1. Choose Draw → Modeling → Pyramid.
2. At the Specify center point of base or [Edge/Sides]: prompt, enter **S** and press Enter.
3. At the Enter number of sides <3>: prompt, enter **6** and press Enter.
4. At the Specify center point of base or [Edge/Sides]: prompt, enter **0,0** and press Enter.



If you know the length of one of the edges, enter **E** and specify the length. The radius of the base is calculated automatically.

5. At the Specify base radius or [Inscribed] <38.8150>: prompt, enter **80** and press Enter to specify the size of the base for the pyramid.
6. At the Specify height or [2Point/Axis endpoint/Top radius] <15.0000>: prompt, enter **T** and press Enter to specify the top radius for the pyramid.
7. At the Specify top radius <82.6851>: prompt, enter **75** and press Enter.
8. At the Specify height or [2Point/Axis endpoint] <15.0000>: prompt, enter **30** and press Enter.

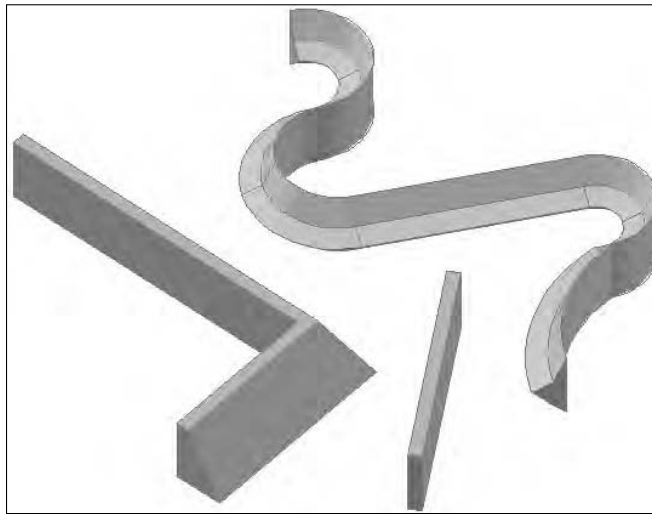
The pyramid tapers towards the top. This is because the base is larger than the radius for the top of the pyramid. If you enter **0** (zero) for the top radius, the pyramid forms a point.

9. Save the drawing with the name `Completed8-3.dwg` to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.



## Polysolid

*Polysolids* are similar in appearance and functionality to 2D polylines with both width and thickness. But a 2D polyline with width and thickness is still not a 3D solid. Like 2D polylines, polysolids are made up of a combination of straight and curved segments. When you create a polysolid, you can specify points to draw straight or curved segments, or you can select an object. You can select a line, 2D polyline, 2D spline, arc, or circle to create a polysolid from. When you select an object or specify points to create a polysolid, you can specify the width, height, and justification values to use. Figure 8-6 shows several examples of polysolids, from a straight hollow tube to a U-channel.



**Figure 8-6:**  
Different  
objects  
created  
from  
polysolids.

You can create a polysolid with any of the following methods:

- ✓ Choose Draw⇒Modeling⇒Polysolid.
- ✓ Click Polysolid either on the Modeling toolbar or on the 3D Make panel on the Dashboard.
- ✓ Enter **polysolid** at the command prompt.

---

### Exercise 8-4: Creating a 3D Polysolid

For this exercise, open the drawing file `Exercise8-4.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D polysolid.

1. Choose Draw⇒Modeling⇒Polysolid.
2. At the Specify start point or [Object/Height/Width/Justify] <Object>: prompt, enter **H** and press Enter.
3. At the Specify height <1.0000>: prompt, enter **18** and press Enter.
4. At the Specify start point or [Object/Height/Width/Justify] <Object>: prompt, enter **W** and press Enter.

5. At the Specify width <1.0000>: prompt, enter **3** and press Enter.
6. At Specify start point or [Object/Height/Width/Justify] <Object>: prompt, enter **J** and press Enter.
7. At the Enter justification [Left/Center/Right] <Left>: prompt, enter **L** and press Enter.
8. At the Specify start point or [Object/Height/Width/Justify] <Object>: prompt, press Enter and select the 2D polyline.
9. Turn on the Parts layer.  
Click the light-bulb icon next to the Parts layer on the Layers drop-down list on the Layers toolbar or in the Layers Properties Manager (Format↔Layer).
10. Save the drawing with the name Completed8-4.dwg to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.

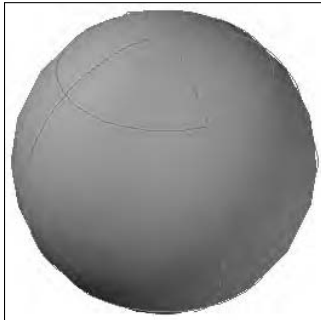


## Solids with Curves

Not all 3D models are created with solids that have straight edges, like boxes or pyramids. AutoCAD comes with several 3D solid primitives that have curved surfaces. Curved solids allow you to make features like holes in mounting plates or spacers between mechanical fasteners.

### Sphere

A *sphere* is a ball-like object that is defined like a circle (see Figure 8-7). You can define a sphere by specifying a center point and then a radius or diameter value. You might use a sphere for the balls in a ball bearing or the end on a hexagon cap nut. Because you create a sphere based on its center point, half the sphere is created above and below the current working plane or the specified point.



**Figure 8-7:**  
A sphere.

You create a sphere in any of the following ways:

- ✓ Choose Draw↔Modeling↔Sphere.
- ✓ Click Sphere either on the Modeling toolbar or on the 3D Make panel on the Dashboard.
- ✓ Enter **sphere** at the command prompt.

### Exercise 8-5: Creating a 3D Solid Sphere

For this exercise, open the drawing file `Exercise8-5.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D solid primitive of a sphere.

1. Choose `Draw` → `Modeling` → `Sphere`.
2. At the `Specify center point or [3P/2P/Ttr]:` prompt, enter the coordinate **10,10**.

You can specify the size of the sphere using two or three points by entering **2P** or **3P** respectively at the command prompt. Enter **T** for the `Ttr` option to define the location that the sphere should be created at based on two tangent points and a radius value.

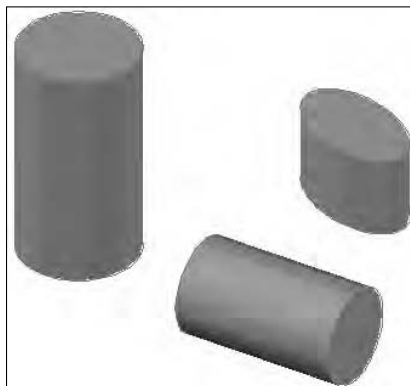
3. At the `Specify radius or [Diameter] <32.0000>:` prompt, enter **16** and press `Enter`.

You can enter **D** to specify the diameter of the sphere.

4. Save the drawing with the name `Completed8-5.dwg` to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Cylinder

A *cylinder* is a long, circular object that's similar to a circle with thickness. You can define a cylinder by specifying a center point for the base, the radius or diameter of the base, and its height. A cylinder can be circular or elliptical (see Figure 8-8). A cylinder might be a shaft that connects a sprocket to a motor, a tool for punching a hole into a mounting plate, a hub of a pulley, or even a roller for a conveyor belt.



**Figure 8-8:**  
Variations of  
circular and  
elliptical  
cylinders.



2. How is a cylinder different from a circle with thickness?
  - A. A circle can be edited with grips, and a cylinder can't be.
  - B. Nothing; they're the same thing.
  - C. A cylinder is capped on the top and bottom.

You start a cylinder with any of these methods:

- ✓ Choose Draw⇨Modeling⇨Cylinder.
- ✓ Click Cylinder either on the Modeling toolbar or on the 3D Make panel on the Dashboard.
- ✓ Enter **cylinder** at the command prompt.

---

### Exercise 8-6: Creating a 3D Solid Cylinder

For this exercise, open the drawing file `Exercise8-6.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D solid primitive of a cylinder.

1. Choose Draw⇨Modeling⇨Cylinder.
2. At the Specify center point of base or [3P/2P/Ttr/Elliptical]: prompt, enter the coordinate **15,15**.

You can specify the size of the cylinder using two or three points by entering **2P** or **3P**, respectively, at the command prompt. Enter **T** for the Ttr option to define the location of the base for the cylinder based on two tangent points and a radius value. Enter **E** for the Elliptical option to create an elliptical cylinder instead of the default circular shape.

3. At the Specify base radius or [Diameter] <7.0000>: prompt, enter **6** and press Enter.

You can enter **D** to specify the diameter for the base of the cylinder.

4. At the Specify height or [2Point/Axis endpoint] <16.0625>: prompt, enter **22.125** and press Enter.

You can determine the height of the cylinder by specifying a point in the drawing or two points after entering **2P** for the two-point option. Alternatively, you can enter **A** for the Axis endpoint option to rotate the cylinder using the center point of its base as the rotation point and to specify its height at the same time.

5. Save the drawing with the name `Completed8-6.dwg` to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.

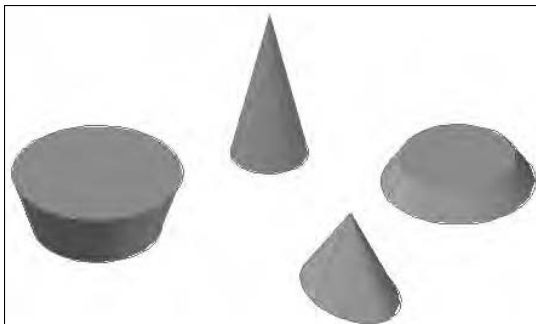
## Cone

*Cones* are objects that have either a peak or a flat top with a circular or elliptical base (see Figure 8-9). When you define a cone, you specify the center point of the base, the radius or diameter of the base, and its height and/or top radius. You can use a cone to model the end of a grease fitting or the head of a screw.

---

**Figure 8-9:**  
Cones can have circular and elliptical bases, like cylinders.

---



You create a cone by following any of these commands:

- ✓ Choose Draw⇒Modeling⇒Cone.
- ✓ Click Cone either on the Modeling toolbar or on the 3D Make panel on the Dashboard.
- ✓ Enter **cone** at the command prompt.

---

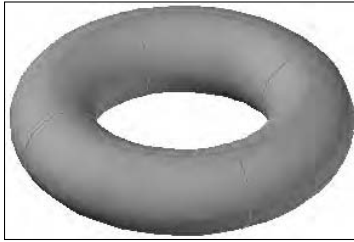
### Exercise 8-7: Creating a 3D Solid Cone

For this exercise, open the drawing file `Exercise8-7.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D solid primitive of a cone.

1. Choose Draw⇒Modeling⇒Cone.
2. At the Specify center point of base or [3P/2P/Ttr/Elliptical]: prompt, enter the coordinate **15,15**.  
  
You can specify the size of the cone using two or three points by entering **2P** or **3P**, respectively, at the command prompt. Enter **T** for the Ttr option to define the location of the base for the cone based on two tangent points and a radius value. Enter **E** for the Elliptical option to create an elliptical base for the cone instead of the default circular shape.
3. At the Specify base radius or [Diameter] <7.0000>: prompt, enter **6** and press Enter.  
  
You can enter **D** to specify the diameter for the base of the cone.
4. At the Specify height or [2Point/Axis endpoint/Top radius] <22.1250>: prompt, enter **T** and press Enter.  
  
You can determine the height of the cone by specifying a point in the drawing or two points after entering **2P** for the two-point option, or entering **A** for the Axis endpoint option to rotate cone using the center point of its base as the rotation point and to specify its height at the same time.
5. At the Specify top radius <0.0000>: prompt, enter **8** and press Enter.
6. At the Specify height or [2Point/Axis endpoint] <22.1250>: prompt, enter **6** and press Enter.
7. Save the drawing with the name `Completed8-7.dwg` to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Torus

A *torus* is a doughnut-shaped object and is one of the least used 3D solid primitives that AutoCAD offers (see Figure 8-10). As you define a torus, you specify the center point of the object, the outer radius or diameter of the torus at its equator, and then the radius or diameter of the tube. A torus might be used as a rubber spacer or to create a raised area around a hole.



**Figure 8-10:**  
A torus.



3. How is a torus similar to a sphere in the way they're created?
- 
- 

You create a torus in the following ways:

- ✓ Choose Draw⇨Modeling⇨Torus.
- ✓ Click Torus either on the Modeling toolbar or on the 3D Make panel on the Dashboard.
- ✓ Enter **torus** at the command prompt.

---

### Exercise 8-8: Creating a 3D Solid Torus

For this exercise, open the drawing file `Exercise8-8.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D solid primitive of a torus.

1. Choose Draw⇨Modeling⇨Torus.
2. At the `Specify center point or [3P/2P/Ttr]:` prompt, enter the coordinate **15,15**.  
 You can specify the size of the torus using two or three points by entering **2P** or **3P**, respectively, at the command prompt. Enter **T** for the Ttr option to define the location of the base for the torus based on two tangent points and a radius value.
3. At the `Specify base radius or [Diameter] <7.0000>:` prompt, enter **6** and press Enter.  
 You can enter **D** to specify the diameter of the torus.
4. At the `Specify tube radius or [2Point/Diameter] <4.0000>:` prompt, enter **2** and press Enter.  
 You can enter **D** to specify the diameter of the tube for the torus or **2P** to specify the size of the tube based on two points.
5. Save the drawing with the name `Completed8-8.dwg` to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Nothing Complex about These Solids

Creating complex solids doesn't require an advanced math degree; they just require some additional input. You create complex solids by taking two or more solids and performing a union operation on them to create a single 3D solid, or you can take a closed 2D object and sweep it along a path to create a 3D solid, like a bent hose or tube. Complex solids form the final model.

You find out about the EXTRUDE command in Chapter 6, which allows you to take an open or a closed 2D object to create a 3D solid or surface. In this section, you create complex 3D solids from primitives and closed 2D objects (profiles).

### *Add some of this and remove a little of that*

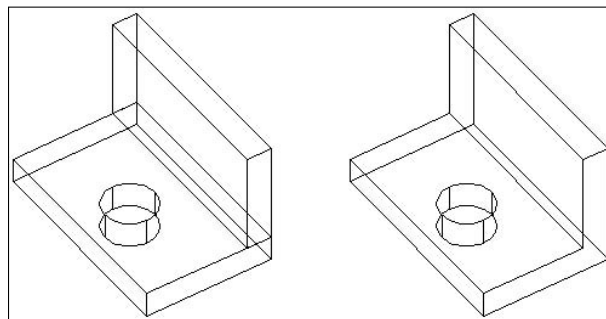
Regions (see Chapter 6) and 3D solids are very close to each other in the way they can be combined or subtracted from each other to create a new complex object. You can use the same commands that you use on regions on 3D solids — UNION, SUBTRACT, and INTERSECT. The INTERSECT command makes a lot more sense when working with 3D solids than it does with regions, though. AutoCAD also provides the INTERFERE command, which is similar to the INTERSECT command, to create a new 3D solid based on how selected 3D solids intersect.



The original objects that you select when using the UNION, SUBTRACT, and INTERSECT commands aren't retained when you create complex 3D solids. If you want to retain the original objects before creating a complex 3D solid with one of these commands, create a copy of the objects in the drawing first. After you create a copy of the objects, perform the desired operation on the 3D solids. When revolving, sweeping, or lofting objects, you can retain the original objects by setting the system variable DELOBJ to the value of 0 (zero).

### *One solid plus one solid equals a new solid*

Creating primitive 3D objects is an important part of modeling in AutoCAD, but creating primitives alone doesn't allow you to calculate values such as mass or the centroid of a part. You must combine solids in order to determine these values. The UNION command (Modify → Solid Editing → Union) combines two or more 3D solids into a single object. Figure 8-11 shows an example of a 3D solid that was created by unioning two solids together to form a single complex solid. You can tell the solids have been unioned together in the 3D solid on the right by the missing edge lines in the lower-right corner and along the back side of the 3D solid.



**Figure 8-11:**  
Combining  
3D solids is  
as easy as  
1, 2, 3.

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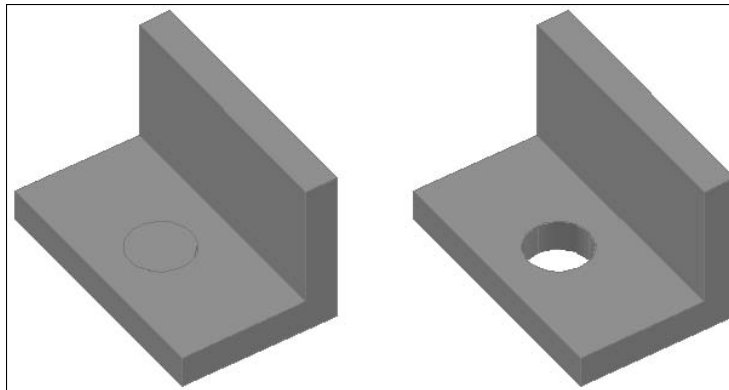
**Exercise 8-9: Adding 3D Solids Together**

For this exercise, open the drawing file `Exercise8-9.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice unioning two 3D solids together to create a new 3D solid.

1. Choose `Modify`→`Solid Editing`→`Union`.
2. Select the two boxes in the drawing.  
The two boxes are separate objects.
3. Press `Esc` to remove the grips from the selected objects.
4. At the `Select objects:` prompt, select the two boxes and press `Enter`.
5. Select the box in the drawing.  
The two boxes are now one single object.
6. Save the drawing with the name `Completed8-9.dwg` to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.

***Taking away from a solid***

Subtracting from a 3D solid is the opposite of adding to one. You don't use the `TRIM` command to remove part of a 3D solid for features like holes or indents. The `SUBTRACT` command (`Modify`→`Solid Editing`→`Subtract`) removes parts of selected 3D solids that intersect another 3D solid. Figure 8-12 shows a 3D solid before and after a cylinder was subtracted from it.



**Figure 8-12:** Subtracting from a 3D solid is as easy as adding to one.

---

**Exercise 8-10: Subtracting a 3D Solid from Another 3D Solid**

For this exercise, open the drawing file `Exercise8-10.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice subtracting a 3D solid from another 3D solid.

1. Choose `Modify`→`Solid Editing`→`Subtract`.
2. At the `Select solids and regions to subtract from .. Select objects:` prompt, select the L-shaped object and press `Enter`.

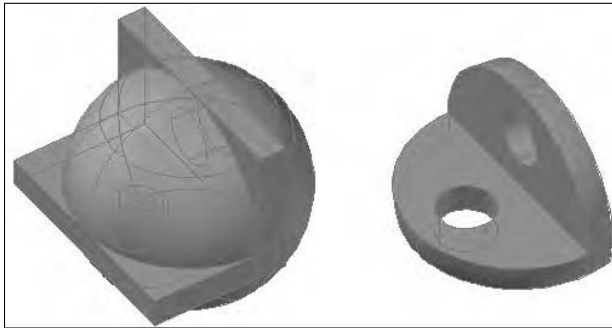


3. At the `Select solids and regions to subtract .. Select objects:` prompt, select the cylinder in the lower portion of the model and press Enter.  
A hole is formed in the L-shaped object.
4. Save the drawing with the name `Completed8-10.dwg` to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.

### ***When solids intersect***

It's possible to create a new 3D solid based on the area that two 3D solids have in common; essentially, it's the area that the 3D solids intersect. The INTERSECT command (Modify→Solid Editing→Intersect) keeps the shared areas of two solids that are used to sculpt and alter the shape of a 3D solid to create new contours or features. Figure 8-13 shows two 3D solids that intersect and the result of using the INTERSECT command on them.

**Figure 8-13:**  
Creating a new 3D solid based on the intersection of two 3D solids.



---

### **Exercise 8-11: Creating a New 3D Solid through the Intersection of Two 3D Solids**

For this exercise, open the drawing file `Exercise8-11.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D solid from the intersection of two 3D solids.

1. Choose `Modify→Solid Editing→Intersect`.
2. At the `Select objects:` prompt, select the two 3D solids in the drawing and press Enter.  
A new 3D solid with curved edges is created.
3. Change the current visual style to `Conceptual`.  
To change the visual style to `Conceptual`, select `Conceptual` from the Visual Styles drop-down list on the Dashboard or choose `View→Visual Styles`.
4. Save the drawing with the name `Completed8-11.dwg` to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.

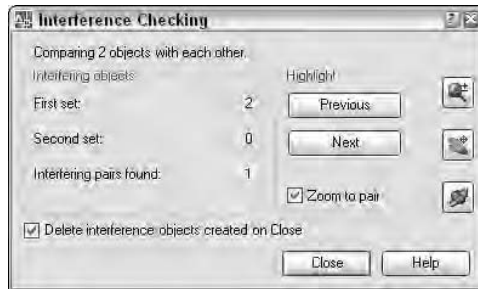
### ***Interference!!!***

Checking for *interference* between two solids is a common practice when working with 3D models to make sure there is clearance between parts. Interference between two or more solids is when two or more solids intersect or cross through each other (not when

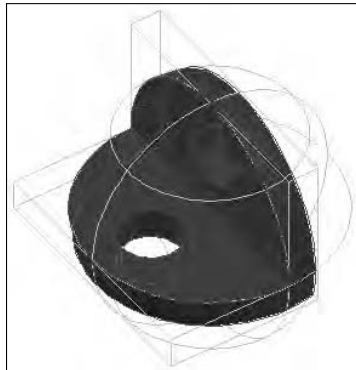
edges or faces touch). The INTERFERE command (Modify→Solid Editing→Interference Checking) checks to see whether 3D solids intersect, and it's similar to the INTERSECT command with a few differences. By default, the INTERSECT command deletes the original objects, whereas the INTERFERE command retains the original objects and creates a new 3D object in the location where the 3D solids intersect.

The INTERFERE command allows you to see the new 3D solids that could be created based on the intersecting areas of the selected 3D solids. After you select the pairs of 3D solids to check, the Interference Checking dialog box (see Figure 8-14) is displayed. The Interference Checking dialog box allows you to step through each of the different sets of the 3D solids that you selected. By finding issues with intersecting 3D solids during the design process, you avoid lost time when creating prototypes or even during production. Figure 8-15 shows the area where two 3D solids intersect in a shaded view.

**Figure 8-14:**  
You can use the Interference Checking dialog box to call attention to interference between 3D solids.



**Figure 8-15:**  
Using INTERFERE to detect issues in a design.



### Exercise 8-12: Creating a New 3D Solid Based on the Interference of Two 3D Solids



For this exercise, open the drawing file `Exercise8-12.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D solid from the intersection of two 3D solids with the INTERFERE command.

1. Choose Modify→Solid Editing→Interference Checking.
2. At the `Select first set of objects or [Nested selection/Settings]:` prompt, select the sphere and the L-shaped objects in the drawing and press Enter.

3. At the `Select second set of objects or [Nested selection/check first set]<check>:` prompt, press `Enter` to accept the default option to check the selected objects to see whether they intersect.

The Interference Checking dialog box is displayed, and the area that the two 3D solids have in common is highlighted in red.

4. In the Interference Checking dialog box, deselect `Delete Interference Objects Created on Close` and click `Close`.

A new 3D solid is created based on the intersection of the two 3D solids.

5. To see the new 3D solid object, start the `MOVE` command, and when prompted to select an object, enter `L` to select the last object created.

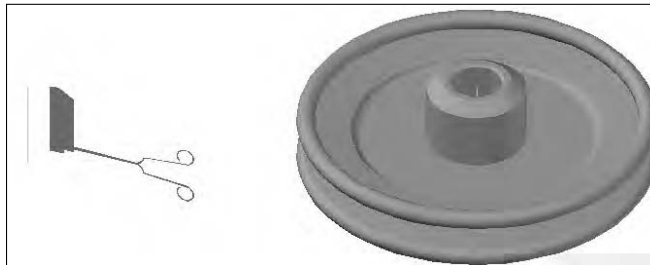
The new 3D solid that was created based on the intersection of the two 3D solids is highlighted. Move the 3D solid away from the center of the sphere and the L-shaped object.

6. Save the drawing with the name `Completed8-12.dwg` to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.

## As the model revolves

Extruding 2D objects is great for creating an object with thickness or having an object follow a path, but you can revolve a 2D object around an axis to create circular 3D solids. Figure 8-16 shows the results of revolving a region around a line to create a 3D solid. You can revolve closed 2D objects to create hubs, pulleys, or even shafts.

**Figure 8-16:**  
A region  
revolved  
around an  
axis.



By default, when you use the `REVOLVE` command, the closed 2D objects that you select to revolve are deleted. If you want to retain the closed 2D objects used to create the revolved 3D solid, set the system variable `DELOBJ` to a value of 0 (zero). You can change the value of `DELOBJ` by entering **delobj** at the command prompt and then entering **0** (zero).

You can revolve a closed 2D object to create a 3D solid with the following methods:

- ✓ Choose `Draw` → `Modeling` → `Revolve`.
- ✓ Click `Revolve` either on the Modeling toolbar or on the 3D Make panel on the Dashboard.
- ✓ Enter **revolve** at the command prompt.



### Exercise 8-13: Revolving a Closed 2D Object to Create a 3D Solid

For this exercise, open the drawing file `Exercise8-13.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D solid by revolving a closed 2D object around an axis.

1. Choose `Draw` → `Modeling` → `Revolve`.
2. At the `Select objects to revolve:` prompt, select the closed 2D object and press `Enter`.
3. At the `Specify axis start point or define axis by [Object/X/Y/Z] <Object>:` prompt, press `Enter`.

You can also pick two points to determine the axis that the closed 2D object should be revolved around, or you can enter one of the options X, Y, or Z to specify the axis of the revolution.

4. At the `Select an object:` prompt, select the line.
5. Specify angle of revolution or, at the `[Start angle] <360>:` prompt, press `Enter` to revolve the closed 2D object a full 360 degrees.

You can also specify how far the object should be revolved around the axis.

6. Turn off the Construction Geometry.

The 3D solid that you create should look like a pulley.

Click the light-bulb icon next to the Construction Geometry layer on the Layers drop-down list on the Layers toolbar or in the Layers Properties Manager (`Format` → `Layer`)

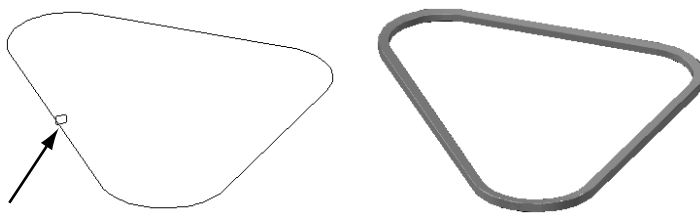
7. Save the drawing with the name `Completed8-13.dwg` to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.



## Sweeping

Sweeping in AutoCAD has nothing to do with getting rid of dust bunnies in the corner of your drawing. Rather, *sweeping* is a technique that you can use to create complex 3D solids. When you create a 3D solid by sweeping, you select a closed 2D object (or profile) and send it along a path. The path that you select can be either an open object such as a line or helix or a closed object such as a circle or closed polyline. You can sweep a 2D object to create a tube that follows a specific angle, the threads on a mechanical fastener, or even a drive belt that runs around several pulleys. Figure 8-17 shows a closed 2D object and a path that is used to create a 3D model of a drive belt.

**Figure 8-17:** No dust pan required for sweeping in AutoCAD.





By default, when you use the SWEEP command, the closed 2D objects that you select to sweep are deleted. If you want to retain the closed 2D objects used to create the swept 3D solid, set the system variable DELOBJ to a value of 0 (zero). You can change the value of DELOBJ by entering **delobj** at the command prompt and then entering **0** (zero).

You can sweep a closed 2D object to create a 3D solid with the following methods:

- ✓ Choose Draw⇒Modeling⇒Sweep.
- ✓ Click Sweep either on the Modeling toolbar or on the 3D Make panel on the Dashboard.
- ✓ Enter **sweep** at the command prompt.



### Exercise 8-14: Sweeping a Closed 2D Object to Create a 3D Solid

For this exercise, open the drawing file `Exercise8-14.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D solid by sweeping a closed 2D object along a path.

1. Choose Draw⇒Modeling⇒Sweep.
2. At the `Select objects to sweep:` prompt, select the closed 2D object indicated by the arrow in Figure 8-12 (shown earlier) and press Enter.
3. At the `Select sweep path or [Alignment/Base point/Scale/Twist]:` prompt, select the closed 2D polyline.

You can also use one of the options to adjust the way the final 3D solid is created.

4. Turn on the layer Parts and turn off the layer Construction Geometry.

Three pulleys are displayed, and the path that you used to sweep the closed 2D object is hidden.

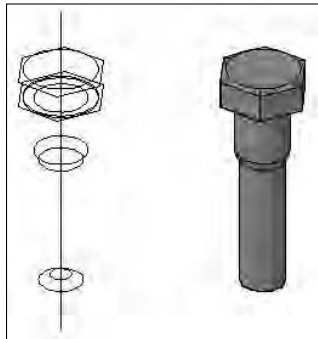


Click the light-bulb icon next to the Parts and Construction Geometry layers on the Layers drop-down list on the Layers toolbar or in the Layers Properties Manager (Format⇒Layer)

5. Save the drawing with the name `Completed8-14.dwg` to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Lofting around

No, lofting isn't something you do in your loafers around the house while modeling in AutoCAD. *Lofting* is a 3D modeling technique that allows you to create 3D solids based on cross sections created from closed 2D objects. The order you select the cross sections in affects how the 3D solid is created. Figure 8-18 shows the cross sections and guide used to create the basics of a 3D model of a bolt using lofting. Along with cross sections to define the 3D solid, you can use a path and guides to help define what the final 3D solid will look like. If you use cross sections only, the Loft Settings dialog box (see Figure 8-19) is displayed to help you fine-tune what the lofted 3D solid will look like.



**Figure 8-18:**  
Example of  
a lofted 3D  
solid.



**Figure 8-19:**  
Tuning the  
3D solid  
with the Loft  
Settings  
dialog box.



By default, when you use the LOFT command, the closed 2D objects that you select to loft are deleted. If you want to retain the closed 2D objects used to create the lofted 3D solid, set the system variable DELOBJ to a value of 0 (zero). You can change the value of DELOBJ by entering **delobj** at the command prompt and then entering **0** (zero).

You can loft closed 2D objects to create a 3D solid with the following methods:

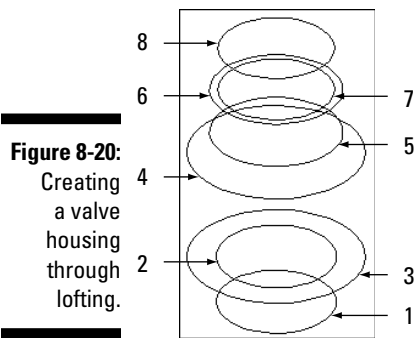
- ✓ Choose Draw→Modeling→Loft.
- ✓ Click Loft either on the Modeling toolbar or on the 3D Make panel on the Dashboard.
- ✓ Enter **loft** at the command prompt.

### Exercise 8-15: Lofting Closed 2D Objects to Create a 3D Solid



For this exercise, open the drawing file `Exercise8-15.dwg` from the Chapter 8 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D solid by using several different cross sections to create a lofted 3D solid.

1. Choose Draw→Modeling→Loft.
2. At the `Select cross sections in lofting order:` prompt, select the cross sections in the order indicated by Figure 8-20.



**Figure 8-20:**  
Creating a valve housing through lofting.

3. After you select all the cross sections, press Enter.
4. At the Enter an option [Guides/Path/Cross-sections only]<Cross-sections only>: prompt, press Enter to choose the default option Cross Sections Only.

The Loft Settings dialog box is displayed. If you have a path or guide to use when lofting, choose the appropriate option.

5. In the Loft Settings dialog box, select Ruled under the Surface Control at Cross Sections.

Select one of the other options if you need to or want to see how these options affect the final lofted object. If the Preview Changes check box is selected, the lofted 3D solid changes in the drawing in real time as you make changes in the Loft Settings dialog box.

6. Click OK to create the lofted 3D solid.
7. Use the 3DORBIT command to view the lofted object.

To start one of the 3D orbiting commands, choose View↔Orbit and then one of the items under the Orbit submenu.

Hold down the Shift key and middle mouse button to enter transparent orbiting.

8. Save the drawing with the name Completed8-15.dwg to the Chapter 8 folder in the My Practice Drawings folder under My Documents and close the drawing.



## Converting objects to 3D solids

Starting with AutoCAD 2007, you can convert 2D objects to 3D solids objects by using the CONVTSOLID command (Modify↔3D Operations↔Convert to Solid). To create extruded 3D solids from 2D objects, select one of the 2D objects compatible with the command. The objects that you can select must be closed and have thickness assigned to them. The CONVTSOLID command allows you to convert the following objects to extruded 3D solid objects:

- ✓ Circle with thickness
- ✓ 2D polyline with thickness and no width
- ✓ 2D polyline with thickness and uniform width

You can also convert 2D objects into 3D surfaces by using the CONVTSURFACE command. To discover more about the CONVTSURFACE command, see Chapter 13.

## *Answers to Practice Opportunities*

**1. When is a wedge useful?**

A. Wedges are helpful when you need an inclined plane or a shim because a wedge is a sloped box that has no height on one end and a specified height on the other end, which allows for a transition from one plane to another. Answer B is wrong because the edges of a slot for a mechanical fastener like a screw don't converge to a sharp edge, and C is not true because wedges are created from straight edges and not curved edges.

**2. How is a cylinder different from a circle with thickness?**

C. A cylinder is capped at the top and bottom, whereas a circle with thickness is not. Answer A is wrong because both circles and 3D solid cylinders can be grip-edited even though they can be edited in different ways. Answer B is also incorrect because they're different types of objects.

**3. How is a torus similar to a sphere in the way they're created?**

Just like the sphere, a torus is created half above and half below the center point specified.



## Chapter 9

# Modifying and Analyzing Solids

### *In This Chapter*

- ▶ Modifying solids
- ▶ Obtaining information from a model

The preceding chapter tells you about primitive and complex 3D solids and how to create them. After you have created 3D primitives or complex 3D solids, you most likely have to modify the 3D solids so your 3D model has the final look it needs. When modifying 3D solids, you can add features like fillets and bevels, or you can even change the shape of a 3D solid that was used to make a complex object if the dimensions of a part change. The exercises in this chapter show you how to modify primitive and complex 3D solids and to get some information from the 3D model to help ensure that the design is safe for its application.

## *Modifying 3D Solids*

Creating 3D solids is only part of the puzzle when modeling in AutoCAD. After you create a 3D solid, whether it's primitive or complex, you most likely still have to modify the object in some way to create the final modeled part. In Chapter 6, I cover some of the basic commands that you can use to modify both 3D solids and surfaces: 3DMOVE, 3DROTATE, MIRROR3D, ALIGN, 3DALIGN, and 3DARRAY. I also cover the commands UNION, SUBTRACT, INTERSECT, and INTERFERE in Chapter 8, which allow you to modify 3D solids.

You can modify a 3D solid in other ways:

- ✓ The Properties palette allows you to modify some of the dimensional properties of 3D solid primitives.
- ✓ Grip editing allows you to modify a 3D solid in different ways. The type of grips that are displayed for the 3D solid depends on whether it is a primitive or complex 3D solid.
- ✓ You can select the subobjects of a 3D solid (edge, face, or vertex) and change their properties using the Properties palette or grips.
- ✓ Add fillets and chamfers by using the FILLET and CHAMFER commands.
- ✓ Slice a 3D solid so you can dimension parts of the model or to show some of the internal features of a model with the SLICE command.
- ✓ The SOLIDEDIT command gives you the ability to manipulate the faces, edges, or body of a 3D solid in many different ways. With the SOLIDEDIT command, you can assign materials for rendering the model (see Chapter 17 for more on materials) and move or rotate a face on a solid.

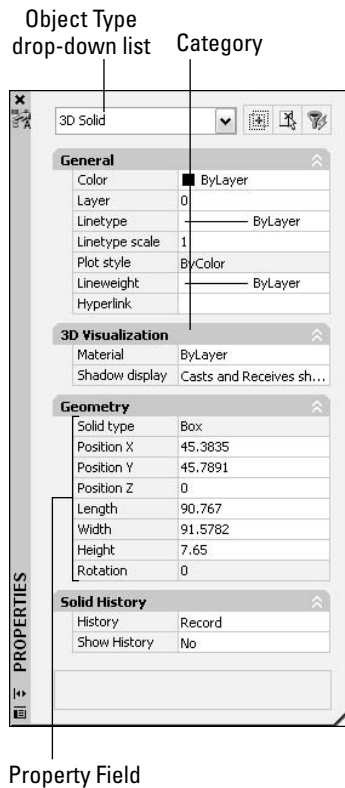
- ✓ You can imprint 2D objects onto the face of a 3D solid so you can extrude geometry to create new complex 3D solids or to create new faces to add materials, among other options, with the IMPRINT command.
- ✓ Extrude closed boundaries to create new 3D solids or to modify the face on a 3D solid with the PRESSPULL command.



1. The following exercises show you how to modify 3D solids, specifically a box and a cylinder. What menu command do you choose to create a box or cylinder?

## Properties palette

You can use the Properties palette (see Figure 9-1) to change the dimensional properties of 3D solid primitives and other settings that can affect the visual appearance of a model or the way you can edit 3D solids. The Properties palette is a modeless dialog box that allows you to modify the properties of a 3D solid in the drawing, such as its height, width, radius, and many other properties that can be modified with or without grips.



**Figure 9-1:**  
The Properties palette.

You can change the properties of an object by selecting the object first and then displaying the Properties palette, or you can display the Properties palette and then select the object you want to modify. Display the Properties palette in the following ways:

- ✓ Choose Tools→Palettes→Properties.
- ✓ Click Properties on the Standard or Standard Annotation toolbar.
- ✓ Select an object and right-click to display an object shortcut menu. Choose Properties from the shortcut menu.
- ✓ Press Ctrl+1.
- ✓ Enter **properties** at the command prompt.

After you select an object, its properties are displayed in different categories on the Properties palette, such as General, 3D Visualization, Geometry, and Solid History. Under each category are specific properties that you can edit or which are read-only for the selected object. You can select more than one object at a time, and the common properties for the selected objects are displayed. If any common properties have different values from each of the selected objects, the text *Varies* is displayed in the field.

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### Exercise 9-1: Modifying 3D Solids Using the Properties Palette

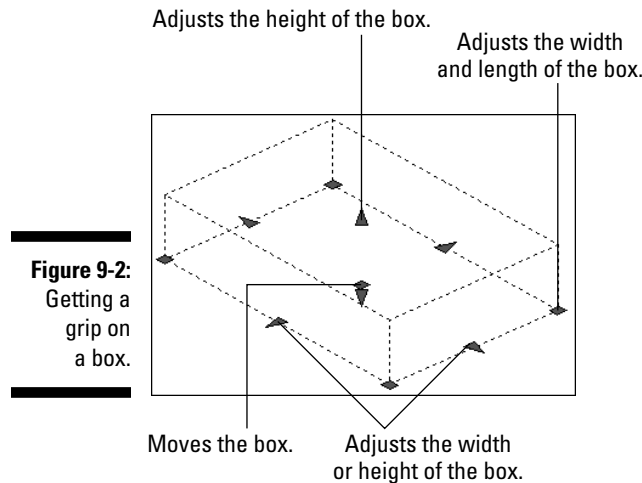
For this exercise, open the drawing file `Exercise9-1.dwg` from the Chapter 9 folder in the My Practice Drawings folder under My Documents. Then practice modifying a box and cylinder 3D solid primitives using the Properties palette.

1. Select the box and right-click.
2. Choose Properties from the shortcut menu to display the Properties palette.
3. On the Properties palette, select the Length field and enter **91**.
4. Select the Width field and enter **93**.
5. Enter **45.5** in the Position X field and **46.5** in the Position Y field.
6. Press Esc and select the cylinder.
7. Enter **56** in the Position X field, **46.5** in the Position Y field, and **7.65** in the Position Z field.
8. Select the Height field and enter **22.35**.
9. Save the drawing with the name `Completed9-1.dwg` to the Chapter 9 folder in the My Practice Drawings folder under My Documents and close the drawing.

## *Grip editing*

Grips are a powerful way to quickly modify 2D objects and an equally powerful way to modify 3D solids. Prior to AutoCAD 2007, it wasn't possible to modify 3D solids using grips (with the exception of moving a 3D solid using a single base grip). Starting with AutoCAD 2007, support for grip editing for 3D solids was added. One of the main changes that allowed for the introduction of grip editing for 3D solids was that the objects retained an understanding of what type of primitive they were or complex 3D solid in the cases where the 3D solid was created through lofting, sweeping, or extruding closed 2D objects.

To use grips to modify a 3D solid, make sure no command is active and select a 3D solid you want to edit. When the grips are displayed for the 3D solid, click one of the grips and drag in the drawing window to modify the 3D solid; click in the drawing to set the new value. Depending on the type of 3D solid you're editing, you get a different number and types of grips. For example, if you select a 3D solid box primitive, 11 grips are displayed (see Figure 9-2). The grips allow you to adjust the height, width, and length of the box along with being able to move the box using its base point.



**Figure 9-2:**  
Getting a  
grip on  
a box.



If the grips aren't displayed for the selected 3D solid, choose Tools⇨Options to display the Options dialog box. In the Options dialog box, click the Selection tab and select the Enable Grips check box under the Grips section. You can also change the color and size for the displayed grips under the Grip Size and Grips sections.



### Exercise 9-2: Modifying 3D Solids with Grips

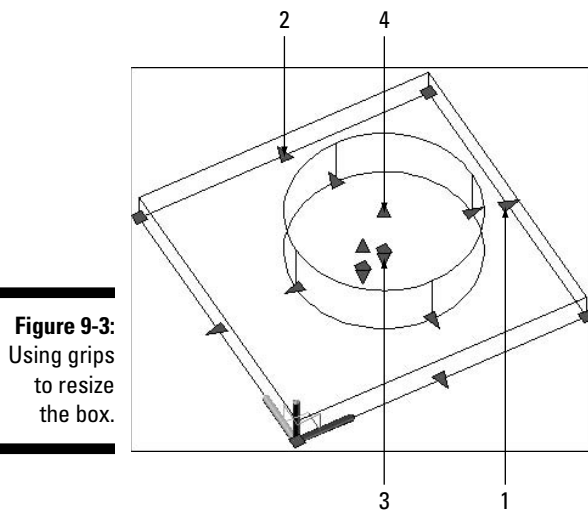
For this exercise, open the drawing file `Exercise9-2.dwg` from the Chapter 9 folder in the My Practice Drawings folder under My Documents. Then practice modifying a box and cylinder 3D solid primitives using grips.

1. Turn on ortho mode, turn off object snap, and turn on dynamic input.



To turn on ortho mode, click ORTHO on the status bar until the message `<Ortho off>` is displayed in the command line window and the ORTHO button appears popped in. To turn off object snap, click OSNAP on the status bar until the message `<Osnap off>` is displayed in the command line window and the OSNAP button appears popped out. To turn on dynamic input, click DYN on the status bar so that the DYN button appears depressed.

2. Select the box to display its grips and select the rightmost grip, indicated by 1 in Figure 9-3.
3. Drag to the crosshairs to the right to establish the use of direct distance entry. Enter `12` and press Enter to change the size of the box 12 units in the X direction.
4. Select the topmost grip, indicated by 2 in Figure 9-3, and drag the crosshairs away from the center of the box.
5. Enter `15` and press Enter to change the size of the box 15 units in the Y direction.
6. Press Esc to clear the grips on the box.



**Figure 9-3:**  
Using grips  
to resize  
the box.

7. Select the cylinder to display its grips and click the square grip at the bottom of the cylinder indicated by 3 in Figure 9-3.
8. At the command prompt, enter **#56,46.5,7.65** and press Enter.  
Using the pound (#) symbol when entering coordinates at the dynamic input tells AutoCAD to use the absolute Cartesian coordinate value. If you omit the # in this case, the coordinates are evaluated as relative coordinate values because dynamic input is turned on. To find out more about the special symbols that you can use for evaluating coordinate values with dynamic input enabled, see the topic *Enter 3D Cartesian Coordinates* in AutoCAD's Online Help system.
9. Select the triangle grip that points upward, located at the center of the cylinder (indicated by 4 in Figure 9-3), and drag the crosshairs towards the top of the drawing window.
10. Enter **7.6** and press Enter to adjust the height of the cylinder.
11. Save the drawing with the name `Completed9-2.dwg` to the Chapter 9 folder in the My Practice Drawings folder under My Documents and close the drawing.

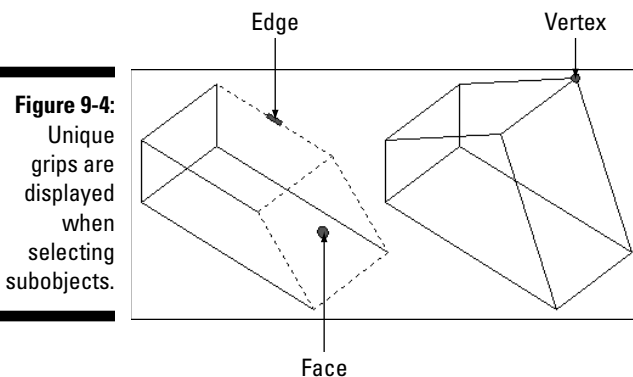
## Selecting a subobject

Selecting 3D solids to modify isn't much different from selecting 2D objects such as lines and arcs, but there are a few unique exceptions. With 3D solids, you can use the standard object selection methods that you've been using since you first started working with AutoCAD, which include implied windowing, fences, single picking, and the other object-selection methods that AutoCAD offers. In addition to the standard object selection methods, you can perform what's known as a *subobject selection*.



2. Which of the following is not a subobject of a 3D solid?
  - A. Face
  - B. Edge
  - C. Intersection
  - D. Vertex

Figure 9-4 shows the elements that make up a 3D solid: face, edge, and vertex. To select a subobject of a 3D solid, you can use the **SU** option at the **Select objects:** prompt of a command by entering **su** at the command prompt and pressing Enter, or you can hold down the Ctrl key when selecting objects. The Ctrl key allows you to select subobjects before you start a command, similar to using the implied window or crossing selection methods. In fact, you can hold down Ctrl and use either the implied window or crossing selection method to select several vertices, edges, or faces at one time. When a subobject is selected, a special grip is displayed that allows you to stretch the selected subobject. Although you can select curved and straight edges on a 3D solid, only straight edges display a grip that allow you to stretch it. Later in this chapter, you find out about the commands **SOLIDEDIT** and **PRESSPULL**, which both take advantage of selecting subobjects to modify a 3D solid.



**Figure 9-4:** Unique grips are displayed when selecting subobjects.

### Exercise 9-3: Selecting Subobjects on a 3D Solid



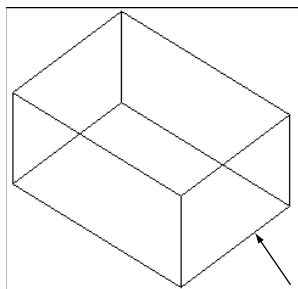
For this exercise, open the drawing file **Exercise9-3.dwg** from the Chapter 9 folder in the My Practice Drawings folder under My Documents. Then practice using subobject selection methods to modify a 3D solid.

1. Turn on Polar Tracking, Snap, and Dynamic Input. Turn off Object Snap.



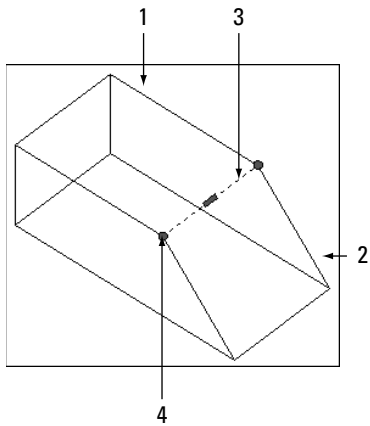
To turn on Polar Tracking, click **POLAR** on the status bar until the message **<Polar on>** is displayed in the command line window and the **POLAR** button appears depressed. To turn on Snap, click **SNAP** on the status bar until the message **<Snap off>** is displayed in the Command Line window and the **OSNAP** button appears depressed. To turn off object snap, click **OSNAP** on the status bar until the message **<Osnap off>** is displayed in the Command Line window and the **OSNAP** button appears popped out.

2. Hold down Ctrl and select the bottom edge of the box, indicated by the arrow in Figure 9-5.



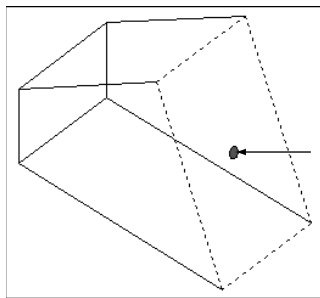
**Figure 9-5:** Editing a box using the edge subobject selection.

3. Click the grip that's displayed and drag the crosshairs away from the center of the box along the polar tracking vector.
4. Click when the tooltip displays `Polar: 25.0000 < 0.`
5. Press Esc to deselect the selected edge.
6. Hold down Ctrl and pick in the drawing window near the locations indicated by 1 and then 2 in Figure 9-6 to use implied windowing to select the vertices of the box.  
When selecting the locations in the drawing window indicated by 1 and 2, make sure not to select an object directly.
7. Hold down Shift and Ctrl and click the edge indicated by 3 in Figure 9-6.
8. Click the grip indicated by 4 in Figure 9-6.



**Figure 9-6:**  
Editing a box using the vertex subobject selection.

9. Drag the crosshairs straight up in the drawing window and click when the tooltip displays `Polar: 35.0000 < +Z.`
10. Press Esc to deselect the selected vertices.
11. Hold down Ctrl and select the face indicated by the arrow in Figure 9-7.



**Figure 9-7:**  
Editing a box using the face subobject selection.

12. Click the grip and drag the crosshairs away from the center of the box along the polar tracking vector.
13. Click when the tooltip displays `Polar: 15.0000 < 0.`
14. Save the drawing with the name `Completed9-3.dwg` to the Chapter 9 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Working with the history of a 3D solid

As you modify 3D solids, AutoCAD is busy taking notes in the background that you might not even know are there. AutoCAD is creating a running history of the objects that are used to create complex 3D solids when using the commands UNION, SUBTRACT, INTERSECT, PRESSPULL, FILLET, and CHAMFER. (I talk about the UNION, SUBTRACT, and INTERSECT commands in Chapter 8, and I cover the commands PRESSPULL, FILLET, and CHAMFER later in this chapter.)

A 3D solid's history comes in handy when you need to modify the 3D solid. For example, if you subtracted a cylinder from a 3D solid to create a hole and later find out that it's in the wrong location and/or is the wrong size, you can use the history to change the size of the cylinder or its location. If you change the cylinder, the 3D solid that you subtracted the cylinder from is updated automatically without your having to first add to the solid and then subtract the new cylinder again.



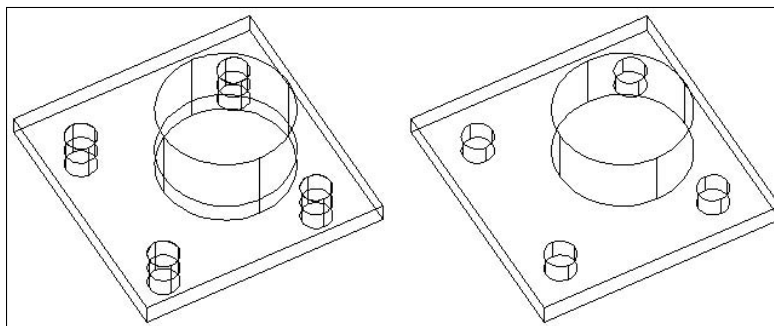
The SOLIDEDIT command (covered later in this chapter) doesn't take advantage of using the history feature. Be careful with which commands you use and in which order you use them if you want to use the history of a 3D solid later to make edits to the original objects.

AutoCAD records the history for any new 3D solids, but you might want to turn on the feature for any 3D solids that were created in a previous release or that the feature was disabled for. The History property under the Solid History category of the Properties palette allows you to control whether the history is retained when modifying a 3D solid. When the History property is set to Record for a 3D solid, the 3D solids are retained with the object to allow you to later modify the original objects.

If you set the value for the History property to None, the previously recorded history is erased, and no new history is created. The BREP command removes the recorded history from a 3D solid and sets its History property to None. If you don't want the history for any of the 3D solids to be retained, use the system variable SOLIDHIST. When SOLIDHIST is set to the value 0 (zero), no history is retained for any 3D solids in the drawing window even if the History property for a 3D solid is set to Record. To change the value of the SOLIDHIST system variable, enter **solidhist** at the command prompt and then enter **0** (zero) at the Enter new value for SOLIDHIST <1>: prompt.

The history that's retained for a 3D solid isn't displayed in the drawing window. To display the history of a 3D solid, either set the Show History property to Yes before modifying a 3D solid or set the system variable SHOWHIST to a value of 2. Figure 9-8 shows a 3D solid with its history displayed on the left and without its history displayed on the right.

**Figure 9-8:**  
A solid model with the original objects displayed and stored in the 3D solid's history.







### Exercise 9-4: Stepping Back in Time to Modify a 3D Solid

For this exercise, open the drawing file `Exercise9-4.dwg` from the Chapter 9 folder in the My Practice Drawings folder under My Documents. Then practice using the history of a complex 3D solid to edit the original box and cylinder used to create the solid.

1. Select the box and display the Properties palette.  
To display the Properties palette, choose `Tools` → `Palettes` → `Properties`.
2. On the Properties palette under the Solid History category, select the History field and select Record from the drop-down list.
3. Union the box and cylinder together.  
To union the two objects to create a single 3D solid, choose `Modify` → `Solid Editing` → `Union`. When the `Select objects:` prompt is displayed, select the two objects and press Enter.
4. Select the 3D solid to verify the two objects are unioned together and press Esc to deselect the 3D solid.
5. At the command prompt, enter `showhist` and press Enter.
6. At the prompt `Enter new value for SHOWHIST <1>:` enter `2` and press Enter.  
The part of the cylinder that extended into the box is now displayed. This is the part of the cylinder that was removed when the two were unioned together.
7. Hold down Ctrl and select the cylinder.  
As the crosshairs pass over the cylinder with Ctrl held down, only the cylinder is highlighted. The grips for the cylinder are displayed so you can now modify the cylinder as needed.
8. Press Esc to clear the grips for the cylinder.
9. Select the 3D solid again and set the History property for the 3D solid to None with the Properties palette.  
The part of the cylinder that was removed during the unioning operation disappears when the history is removed.
10. Hold down Ctrl and try to select the cylinder again.  
This time you can select only the edges and faces of the cylinder and not the original cylinder 3D primitive because the history has been removed from the 3D solid.
11. Save the drawing with the name `Completed9-4.dwg` to the Chapter 9 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Filleting and chamfering

*Fillets* (not filets) and *chamfers* are common features in mechanical designs for parts such as brackets and fasteners. The `FILLET` and `CHAMFER` commands pull double duty in AutoCAD as they allow you to modify both 2D geometry and 3D solids. The process of using the `FILLET` and `CHAMFER` commands on 3D solids is slightly different from modifying 2D geometry, but the end result is the same.

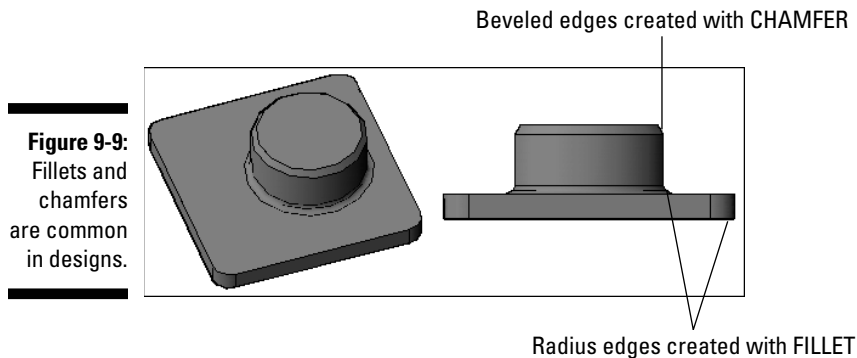
With the FILLET command, you can create radius transitions from one face to the next or to add curved corners on a part. You start the FILLET command in the following ways:

- ✓ Choose Modify→Fillet.
- ✓ Click Fillet either on the Modify toolbar or on the 2D Draw panel on the Dashboard.
- ✓ Enter **fillet** at the command prompt.

The CHAMFER command allows you to create a bevel or angled transition between two faces based on two distance values. You start the CHAMFER command in the following ways:

- ✓ Choose Modify→Chamfer.
- ✓ Click Chamfer either on the Modify toolbar or on the 2D Draw panel on the Dashboard.
- ✓ Enter **chamfer** at the command prompt.

You might use the CHAMFER command to create a taper in a hole so a screw sits flush with the bracket or to add a bevel to the side of a plate. Figure 9-9 shows a plate with curved corners, a radius transition, and a beveled edge.



**Figure 9-9:**  
Fillets and  
chamfers  
are common  
in designs.

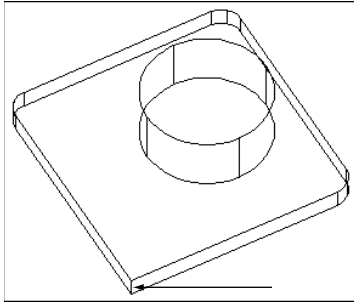
### Exercise 9-5: Adding Fillets and Chamfers

For this exercise, open the drawing file `Exercise9-5.dwg` from the Chapter 9 folder in the My Practice Drawings folder under My Documents. Then practice creating fillets and chamfers on a 3D solid.



1. Choose Modify→Fillet.
2. At the `Select first object or [Undo/Polyline/Radius/Trim/Multiple]:` prompt, select the vertical edge in the lower-left corner of the box, indicated by the arrow in Figure 9-10.
3. At the `Enter fillet radius <1.0000>:` prompt, enter **7.65** and press Enter twice.  
A fillet is added, and it matches the other corners of the mount plate.
4. Press Enter to repeat the FILLET command.
5. At the `Select first object or [Undo/Polyline/Radius/Trim/Multiple]:` prompt, select the bottom edge of the cylinder.

**Figure 9-10:**  
Adding  
fillets and  
chamfers.



6. At the `Enter fillet radius <7.6500>`: prompt, enter **3** and press Enter.
7. Press Enter to fillet only the selected edge.  
A fillet is added and a transition is created between the box on the cylinder.
8. Choose `Modify`→`Chamfer`.
9. At the `Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]`: prompt, select the top edge of the cylinder.
10. At the `Enter surface selection option [Next/OK (current)] <OK>`: prompt, press Enter to accept the top of the cylinder as the base surface.
11. At the `Specify base surface chamfer distance`: prompt, enter **2** and press Enter.
12. At the `Specify other surface chamfer distance <2.0000>`: prompt, enter **3** and press Enter.
13. At the `Select an edge or [Loop]`: prompt, select the top edge of the cylinder again and press Enter.  
A chamfer is added along with a beveled edge around the top of the cylinder.
14. To better see the new fillets and chamfers, switch to the Conceptual visual style and the Left standard view.  
To switch to the Conceptual visual style, choose `View`→`Visual Styles`→`Conceptual`. You can view the model from the Left standard view by choosing `View`→`3D View`→`Left`.
15. Save the drawing with the name `Completed9-5.dwg` to the Chapter 9 folder in the My Practice Drawings folder under My Documents and close the drawing.

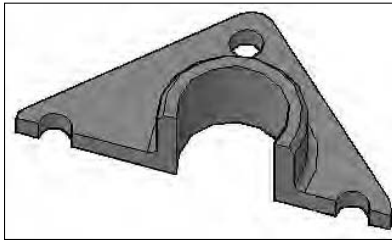


3. To remove fillets and chamfers, enter **solidedit** at the command prompt and choose the \_\_\_\_\_ option under Face.

## *Slicing and dicing 3D solids*

Although AutoCAD doesn't make a good food processor, it does have the ability to slice a 3D solid along a specified plane. The SLICE command carves up a 3D solid along a given plane using a planar object, surface, or three points in addition to a few other options. (I cover planar objects and surfaces in Chapter 13.) When you slice a 3D solid, AutoCAD creates two new solids. From the two solids that are created, you can either choose to keep both halves or keep one of the two new solids that are created.

Slicing a 3D solid allows you to get a better look at the inside of the model for creating a diagram of how the model is assembled with multiple different parts, to create new contours on a 3D solid, to create a section cut view when plotting a hardcopy of the model, or to create a view of the model. The latter is usually done through the use of other features in AutoCAD like live sectioning or flattening. I talk about live sectioning and flattening in Chapter 10. Figure 9-11 shows a 3D solid sliced diagonally to show different features.



**Figure 9-11:**  
A slice of a  
3D solid.

You start the SLICE command in the following ways:

- ✓ Choose Modify → 3D Operations → Slice.
- ✓ Click Slice on the 3D Make panel of the Dashboard.
- ✓ Enter **slice** at the command prompt.

---

### Exercise 9-6: Creating a Slice of Model

For this exercise, open the drawing file `Exercise9-6.dwg` from the Chapter 9 folder in the My Practice Drawings folder under My Documents. Then practice slicing a 3D model.

1. Enabled Object Snaps and make sure that the Center object snap is the only one set.  
Right-click the OSNAP button on the status bar and select Settings from the shortcut menu. In the Drafting Settings dialog box on the Object Snap tab, click Clear All and then click the check box next to Center. If Object Snap On (F3) isn't selected, select it. Click OK to return to the drawing window.
2. Choose Modify → 3D Operations → Slice.
3. At the Select objects to slice: prompt, select the 3D solid and press Enter.
4. At the Specify start point of slicing plane or [planarObject/Surface/Zaxis/View/XY/YZ/ZX/3points] <3points>: prompt, select the center point of the hole in the upper-left corner the model.
5. At the Specify second point on plane: prompt, select the center point of the hole in the lower-right corner the model.
6. At the Specify a point on desired side or [keep Both sides] <Both>: prompt, click in the drawing window above the model to keep only the top half.
7. Save the drawing with the name `Completed9-6.dwg` to the Chapter 9 folder in the My Practice Drawings folder under My Documents and close the drawing.









## Solid editing



The SOLIDEDIT command gives you the capability to modify the faces or edges of a 3D solid in many different ways, along with editing the entire 3D solid known as the *body*. The SOLIDEDIT command offers 16 different ways to modify a 3D solid; some of the options are much more useful than others.






You start the SOLIDEDIT command in the following ways:

- ✓ Choose Modify→Solid Editing and select one of the options from the submenu.
- ✓ Click one of the buttons on the Solid Editing toolbar.
- ✓ Enter **solidedit** at the command prompt.

Table 9-1 lists the options of the SOLIDEDIT command that modify the faces of a 3D solid, and Table 9-2 lists the edge editing options. Table 9-3 lists the 3D solid editing options that apply to the entire body of the 3D solid.

<b>Table 9-1</b>		
<b>Editing Faces</b>		
<b>Button</b>	<b>Command Option</b>	<b>Description</b>
	Extrude	Extrudes the selected face into or away from the 3D solid.
	Move	Relocates the selected face on the 3D solid. You can relocate a hole or slide the side of a 3D solid to change its shape.
	Rotate	Rotates the selected face on the 3D solid.
	Offset	Offsets the selected face on the 3D solid.
	Taper	Tapers the selected face on the 3D solid, which causes the face to flare towards or away from the 3D solid along an axis and the specified angle.
	Delete	Removes the selected face, fillet, or chamfer.
	Copy	Allows you to create a copy of a selected face. A 2D region is created from the copied face.
	Color	Changes the color of the selected face. The color of the face affects how the solid is displayed when a visual style is applied or when a rendering is created.
None	Material	Assigns a material to the face, which is used when the Realistic visual style is applied to the viewport or a rendering is created. I talk about creating and applying materials in Chapter 17. The Material option is available only when using the SOLIDEDIT command at the command prompt.

<b>Table 9-2</b>		
<b>Editing Edges</b>		
<i>Button</i>	<i>Command Option</i>	<i>Description</i>
	Color Edges	Changes the color of the selected edges. The color of the edges affects how the solid is displayed when a visual style is applied.
	Copy Edges	Allows you to create a copy of selected edges, which results in the creation of 2D objects (such as lines and arcs) based on the selected edges.

<b>Table 9-3</b>		
<b>Editing Body</b>		
<i>Button</i>	<i>Command Option</i>	<i>Description</i>
	Imprint	Imprints a 2D or 3D object onto a 3D solid, resulting in the creation of new faces and edges based on the object that is imprinted onto the 3D solid. The object that you want to imprint onto the 3D solid must intersect one of the faces of the 3D solid. After the object is imprinted onto the 3D solid, you can extrude the newly created face to create a new feature like a raised edge or to inset in the face that the imprinted object is on. You can also use the IMPRINT command to perform this task, which I cover in the section “Branding (imprinting) your 3D solids,” later in this chapter.
	Separate Solids	Creates new 3D solids based on each disjointed volume; the option doesn't support the separation of 3D solids that have been modified using Boolean operations.
	Shell	Hollows out a 3D solid based on an offset distance. A positive value creates the wall thickness towards the middle of the 3D solid, whereas a negative value creates the wall thickness to the outside of the 3D solid.
	Clean	Scans a 3D solid for any redundant edges, vertices, and unused geometry and then removes them.
	Check	Validates a 3D solid using the ShapeManager to determine that it's a valid object with no problems.

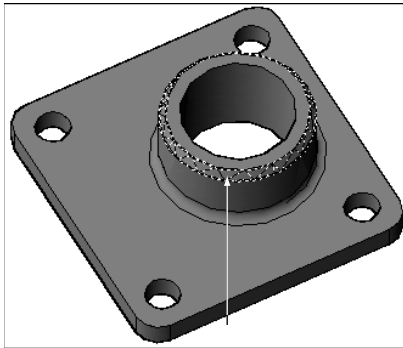
### Exercise 9-7: Deleting a Chamfer

For this exercise, open the drawing file `Exercise9-7.dwg` from the Chapter 9 folder in the My Practice Drawings folder under My Documents. Then practice removing the chamfer from a 3D solid.

1. Choose **Modify**→**Solid Editing**→**Delete Faces**.
2. At the `Select faces or [Undo/remove]:` prompt, select the face indicated by the arrow in Figure 9-12 and press **Enter**.

The chamfer is removed from the 3D solid.

**Figure 9-12:**  
Deleting a  
chamfer  
from a 3D  
solid.



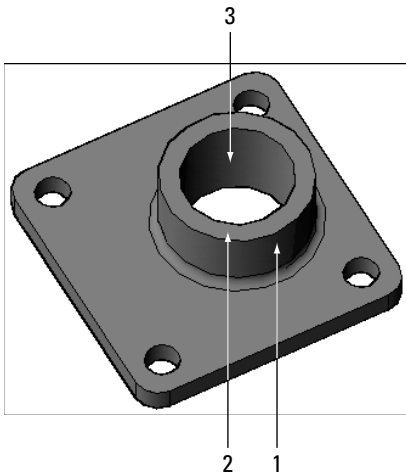
3. Press Enter twice to exit the SOLIDEDIT command.
4. Save the drawing with the name `Completed9-7.dwg` to the Chapter 9 folder in the My Practice Drawings folder under My Documents and close the drawing.

### Exercise 9-8: Tapering a Face

For this exercise, open the drawing file `Exercise9-8.dwg` from the Chapter 9 folder in the My Practice Drawings folder under My Documents. Then practice tapering the face of a 3D solid.

1. Set the center as the only running object snap.
2. Choose `Modify` → `Solid Editing` → `Taper Faces`.
3. At the `Select faces or [Undo/remove]:` prompt, select the face indicated by the 1 in Figure 9-13 and press Enter.

**Figure 9-13:**  
Tapering  
the face of  
a 3D solid.



4. At the `Specify the base point:` prompt, select the center point of the bottom of the cylinder, indicated by 2 in Figure 9-13.
5. At the `Specify another point along the axis of tapering:` prompt, select the center point of the top of the cylinder, indicated by 3 in Figure 9-13.
6. At the `Specify the taper angle:` prompt, enter `5` and press Enter.

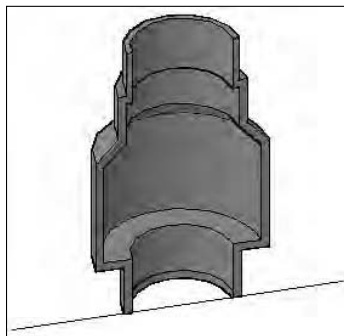
7. Press Enter twice to exit the SOLIDEDIT command.
8. Save the drawing with the name `Completed9-8.dwg` to the Chapter 9 folder in the My Practice Drawings folder under My Documents and close the drawing.



### Exercise 9-9: Shelling Out a 3D Solid

For this exercise, open the drawing file `Exercise9-9.dwg` from the Chapter 9 folder in the My Practice Drawings folder under My Documents. Then practice shelling out a 3D solid.

1. Set the endpoint as the only running object snap.
2. Choose `Modify` → `Solid Editing` → `Shell`.
3. At the `Select a 3D solid:` prompt, select the 3D solid.
4. At the `Remove faces or [Undo/Add/All]:` prompt, press Enter.
5. At the `Enter the shell offset distance:` prompt, enter `3` and press Enter.
6. Press Enter twice to exit the SOLIDEDIT command.
7. Subtract the cylinder that's in the middle of the 3D solid.
8. Choose `Modify` → `3D Operations` → `Slice`.
9. At the `Select objects to slice:` prompt, select the 3D solid and press Enter.
10. At the `Specify start point of slicing plane or [planar Object/Surface/Zaxis/View/XY/YZ/ZX/3points] <3points>:` prompt, select the left end of the line using the endpoint object snap.
11. At the `Specify second point on plane:` prompt, select the right end of the line using the endpoint object snap.
12. At the `Specify a point on desired side or [keep Both sides] <Both>:` prompt, select a point above the 3D solid to keep half of the 3D solid (see Figure 9-14).



**Figure 9-14:**  
Results of  
shelling out  
a 3D solid  
and slicing it  
in half.

13. Save the drawing with the name `Completed9-9.dwg` to the Chapter 9 folder in the My Practice Drawings folder under My Documents and close the drawing.

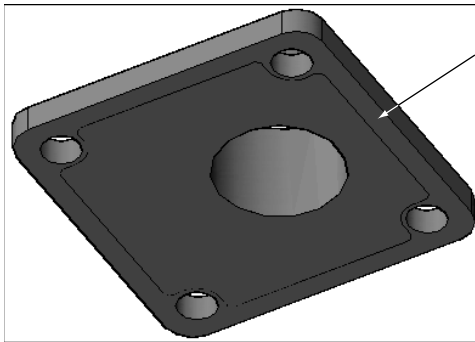


## Branding (imprinting) your 3D solids

You can imprint a 3D solid with a 2D object to create new edges and faces. One purpose of imprinting a 2D object onto a 3D solid is so you can add a different material to the face of a 3D solid. You can also imprint a 2D object onto a 3D solid and then use the Extrude option of the SOLIDEDIT command or the PRESSPULL command (covered later in this chapter) to create a feature like an inset or a hole.

The IMPRINT command imprints a 2D object onto a 3D solid and is just like the Imprint option of the SOLIDEDIT command with the exception that it takes fewer steps to access and use. As with the Imprint option of the SOLIDEDIT command, the 2D object that you want to imprint must intersect a face of the 3D solid. You can use the Dynamic UCS feature to make sure that the 2D object that you want to imprint is drawn right on the face of the 3D solid. (I cover using the Dynamic UCS feature in Chapter 4.) Figure 9-15 shows part of the bottom of a mounting plate drawn with a 2D polyline and imprinted onto a 3D solid.

**Figure 9-15:**  
A 2D  
polyline  
imprinted  
onto the  
face of a  
3D solid.



You can start the IMPRINT command in the following ways:

- ✓ Choose Modify→Solid Editing→Imprint Edges.
- ✓ Click Imprint on the Solid Editing toolbar or 3D Make panel on the Dashboard.
- ✓ Enter **imprint** at the command prompt.

---

### Exercise 9-10: Imprinting a 2D Object onto a 3D Solid

For this exercise, open the drawing file `Exercise9-10.dwg` from the Chapter 9 folder in the My Practice Drawings folder under My Documents. Then practice imprinting a 2D object onto a 3D solid.

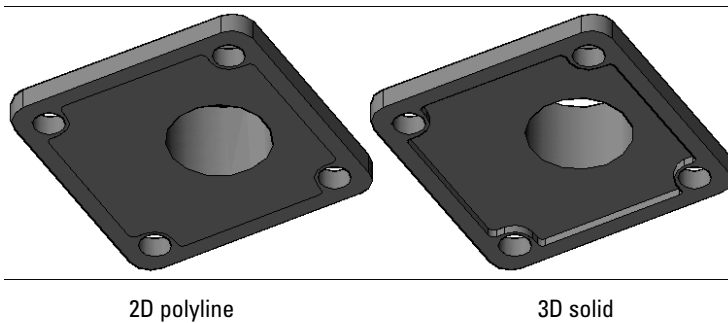
1. Choose Modify→Solid Editing→Imprint.
2. At the `Select a 3D solid:` prompt, select the outside edge of the 3D solid.
3. At the `Select an object to imprint:` prompt, select the 2D polyline toward the middle of the 3D solid.
4. At the `Delete the source object [Yes/No] <N>:` prompt, enter **Y** and press Enter to remove the 2D polyline from the drawing after it is imprinted on the 3D solid.
5. Press Enter to exit the IMPRINT command.

6. Click the 2D polyline.  
The entire 3D solid is selected because the 2D polyline isn't part of the 3D solid.
7. Press Esc to clear the grips and deselect the 3D solid.
8. Save the drawing with the name `Completed9-10.dwg` to the Chapter 9 folder in the My Practice Drawings folder under My Documents and close the drawing.

## *You don't have to get pushy*

The PRESSPULL command was first introduced in AutoCAD 2007 and allows you to skip several steps in the process of editing solids. PRESSPULL combines the work of imprinting and extruding in one step; you can even use the PRESSPULL command to extrude closed boundaries that aren't even on the face of a 3D solid to create a new 3D solid object. Like the command's name suggests, you can create a 3D solid by pulling it out of a closed boundary, or you can press it into a 3D solid in order to perform a subtraction. PRESSPULL gives you a lot of flexibility in creating conceptual models. Figure 9-16 shows the results of pressing a 2D polyline that has been drawn onto a 3D solid to subtract part of the geometry.

**Figure 9-16:**  
Pressing a  
2D polyline  
into a 3D  
solid.



You start the PRESSPULL command in the following ways:

- ✓ Click Presspull either on the Modeling toolbar or on the 3D Make panel on the Dashboard.
- ✓ Hold down Ctrl+Alt and click inside a closed boundary.
- ✓ Enter **presspull** at the command prompt.

### **Exercise 9-11: Modifying a 3D Solid with PRESSPULL**

For this exercise, open the drawing file `Exercise9-11.dwg` from the Chapter 9 folder in the My Practice Drawings folder under My Documents. Then practice pressing a closed boundary into a 3D solid.



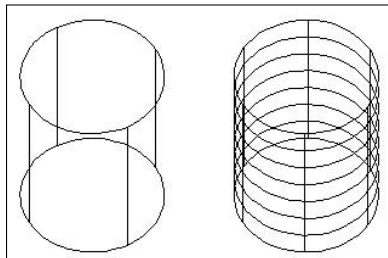
1. Press Esc to make sure that no command is active.
2. Press and hold Ctrl and Alt.
3. Move the crosshairs over the 3D solid and click just to the left of the large hole while still pressing Ctrl and Alt.

4. At the command prompt, type **-3** and press Enter.  
AutoCAD calculates the closed boundary and extrudes it into the 3D solid, resulting in a subtraction.
5. Save the drawing with the name `Completed9-11.dwg` to the Chapter 9 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Exploding solids

3D solids can be exploded like many other types of complex objects in AutoCAD: Simply use the EXPLODE command (Modify→Explode). When you explode 3D solids, they're broken down into surfaces and regions. If the face of a 3D solid is flat, it's most likely converted into a region, whereas a face that has height and is curved is created into a surface. For example, if you explode a cylinder, three new objects are created. Two regions are created for the top and bottom of the cylinder, and a surface is created for the side of the cylinder.

Visually, the new objects look very similar when viewed with a visual style other than 2D wireframe or 3D wireframe. Figure 9-17 shows a primitive 3D solid of a cylinder on the left, but on the right the same 3D solid has been exploded. It's best not to explode a 3D solid because you lose a lot of flexibility in editing the object.



**Figure 9-17:**  
Exploding a  
3D solid.

## Putting Your Model under the Microscope

3D models are great for communicating concepts and the intent of a design, but you can leverage the information contained in the 3D solids of your model for other purposes. AutoCAD provides a few tools to help you collect information about the 3D model in order to help make sure that your designs are safe. However, to get the most design analysis data from your model, you have to obtain an additional software package or two based on the types of parts that you model.

## Getting information from your model

You can use the LIST and MASSPROP commands to get some basic information about 3D solids, such as volume and centroid. The LIST command is rather limited in providing information for 3D solids, except when it comes to primitives. The LIST command displays the length, width, radius, and height among other values of a selected primitive 3D solid. The MASSPROP command displays information that's a little more usable when working with 3D solids, such as moments of inertia, centroid, mass, and several other values. I cover the MASSPROP command in Chapter 6.



4. Open a drawing that contain a 3D solid (use `Exercise9-11.dwg` if you don't have a drawing), activate the LIST command, and then select a 3D solid. What information is displayed if you select a 3D solid when the LIST command is active?

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## *Analyzing your model*

2D drawings are great for communicating your designs to the shop floor, but they don't allow you to perform complex design and engineering analysis on them. When you create a 3D model, you can perform different design and engineering analysis on the model by using additional software. Because AutoCAD is a CAD application that doesn't have a specific target industry, it lacks some tools that might be useful in creating mechanical designs. Similarly, AutoCAD doesn't offer any types of tools for design analysis other than the LIST and MASSPROP commands, which are rather limited in the type of information they offer. Applications such as Autodesk Inventor do a much better job at both modeling and design analysis in one product, and that's because Inventor is designed specifically for mechanical designs. To find out more about Autodesk Inventor, visit the Inventor product page on the Autodesk Web site at [www.autodesk.com/inventor](http://www.autodesk.com/inventor).

Because AutoCAD doesn't provide any real design analysis tools out of the box, you have to rely on other companies for this type of software. The software that you might use varies based on the parts that you plan on modeling and manufacturing. The type of analysis you can perform can vary from stress and fatigue analysis using Finite Element Modeling and Analysis (FEM/FEA) to software that exams how metal or plastic flows to produce an optimized casting or molded part. Other types of software simulate how a part moves or how a complete factory would work. The following is a brief listing of Web sites that have products that can help you analyzing your models if you need to do that as part of your design process.

- ✓ Moldflow Corporation ([www.moldflow.com](http://www.moldflow.com)) has a number of widely used software tools for the injection molding industry that addresses the process from design through to manufacture.
- ✓ EKK, Inc., ([www.ekkin.com](http://www.ekkin.com)) offers CAPCAST for metal casting simulation and analysis.
- ✓ Algor ([www.algor.com](http://www.algor.com)) produces a wide range of FEA software.
- ✓ Nastran ([www.nastran.com](http://www.nastran.com)) produces a wide range of FEA software.

To find out about other tools that you might want to use in analyzing your models, check with your local design software reseller, surf the Internet using your favorite search engine, or check out a trade show that's specific to your industry.

## Answers to Practice Opportunities

**1. What menu command do you choose to create a box or cylinder?**

Choose Draw⇨Modeling⇨Box for a box or Draw⇨Modeling⇨Cylinder for a cylinder and then follow the command prompts. If you need more help to create a box or cylinder, see Chapter 8.

**2. Which of the following is not a subobject of a 3D solid?**

C. Intersection is not a subobject of a 3D solid.

The subobjects that you can manipulate for a 3D solid are

- Face, the surface that is created when edges form a closed boundary, such as the top of a cylinder or the side of a box.
- Edge, the visible lines that are shown on-screen that make up a face.
- Vertex, the location where one or more edges meet. The plural of vertex is *vertices*.

**3. To remove fillets and chamfers, enter solidedit at the command prompt and choose the Delete option under Face.**

**4. What information is displayed if you select a 3D solid when the LIST command is active?**

Your results vary based on the 3D solid that you selected. The following is an example of the information that you might see displayed in the Text Window after selecting a 3D solid with the LIST command:

```
3DSOLID Layer: "0"  
Space: Model space  
Handle = 794  
History = None  
Show History = No  
Bounding Box: Lower Bound X = 0.0000 , Y = 0.0000 , Z =  
0.0000  
Upper Bound X = 91.0000 , Y = 93.0000 , Z = 30.0000
```



## Chapter 10

# Creating Section and Auxiliary Views from Models

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### *In This Chapter*

- ▶ Creating 2D section and auxiliary views from a model
  - ▶ Flattening and creating profiles of a model
- 

**C**hapter 8 tells you all about primitive and complex 3D solids and how to create them. You also practice some basic and advanced modifying techniques on 3D solids in Chapter 9, which include adding to and subtracting from 3D solids and even checking to see whether two or more 3D solids interfere with each other and other techniques. As you go through this chapter, you create section and auxiliary views from your 3D models. The section and auxiliary views allow you to focus on creating your 3D model first and then create standard 2D views to create 2D drawings that explain how a part needs to be manufactured. The exercises in this chapter show you how to bring a 3D model down to 2D objects to help communicate the final design to those in manufacturing who will be tooling the parts and assembling them.

## *Creating Section and Auxiliary Views*

Creating section and auxiliary views from a 3D model can help you better communicate how the model is assembled or even update 2D drawings to ensure they're accurate before manufacturing begins on the parts. Starting with a 2D drawing is common when creating mechanical parts, but when creating 3D models, you might find parts don't align correctly and might require some changes. After you create a 3D model of a part, you can create a 2D view from it that you can use to create or update a set of 2D drawings. AutoCAD comes with different commands that generate 2D representations of a 3D model, and I cover these commands in the following sections.

### *Ready, set, section*

In Chapter 9, I cover the SLICE command, which allows you to cut a 3D solid into two halves along a plane. After you slice a 3D solid along a plane, you can see inside the 3D solid, which helps in designing the part and can give you a better understanding how everything might go together. The downside of the SLICE command is that it modifies a 3D solid. Fortunately, AutoCAD provides two other commands that allow you to create a 2D or 3D representation of a 3D solid without actually modifying the 3D solid: SECTIONPLANE and LIVESECTION.

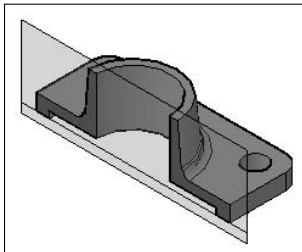
A section plane object determines what part of a 3D model is displayed, similar to how you can slice a 3D model along a plane. You create a section plane object in the following ways:

- ✓ Choose Draw⇒Modeling⇒Section Plane.
- ✓ Click Section Plane on the 3D Make control panel of the Dashboard.
- ✓ Enter **sectionplane** at the command prompt.

When you create a section plane object, you can define it by picking two or more points in the drawing, or you can align the section plane orthogonally to the current UCS.

After you create a section plane object in a drawing, you can enable what is known as *live sectioning*. Live sectioning is a property of a section plane object that allows you to interactively cut through a 3D model as if you used the SLICE command without actually affecting the 3D model. More than one section plane object can exist in a drawing at a time, but only one section plane can have live sectioning enabled in the drawing at a time. Figure 10-1 shows live sectioning enabled on a section plane drawn through a 3D model.

**Figure 10-1:**  
Live sectioning is enabled on a section plane.



You activate live sectioning in the following ways:

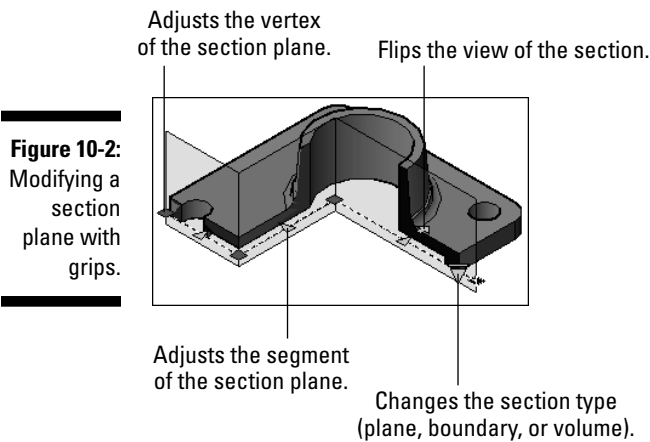
- ✓ Select a section plane object and right-click. From the shortcut menu, choose Activate Live Sectioning.
- ✓ Double-click a section plane object.
- ✓ Set the Live Section property in the Properties palette to Yes.
- ✓ Select a section plane object and right-click. From the shortcut menu, choose Live Section Settings and then select Activate Live Section.
- ✓ Enter **livesection** at the command prompt and select a section plane object.



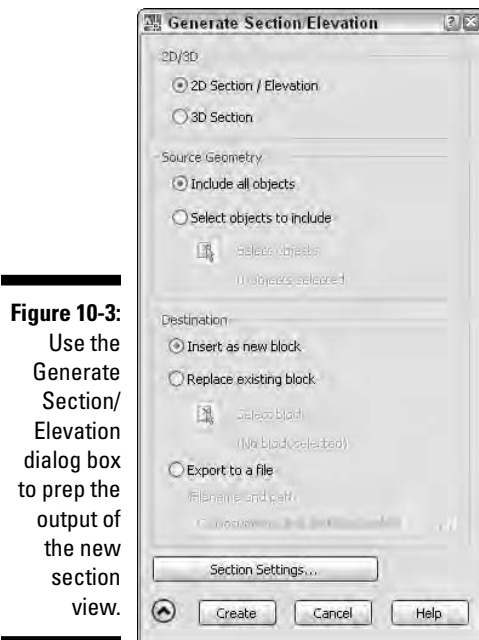
You're able to see part of the 3D model on-screen, but the object itself isn't modified in the same way as using the SLICE command. If you want to, you can choose to see both halves of the 3D model and not just half of the model. Select the section plane with live sectioning enabled and right-click. From the right-click menu, choose Show Cut-Away Geometry. The geometry that's on the opposite side and not normally displayed is shown in red.

You can also adjust the section plane in real time and see the section update. You use grips on the section plane object to control the position and direction of the section plane object. Figure 10-2 shows the grips of a section plane object and what they're used for. Section planes are defined by two or more points to create jogs for a section view if needed. To add a jog to a section plane object, select the section plane and right-click. From the right-click menu, choose Add Jog to Section. Then select the section plane and adjust the jog with the grips that are displayed.



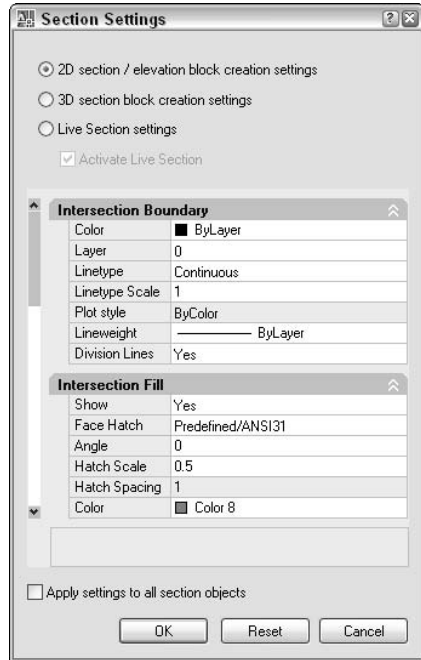


After defining a section plane object, you can create a block made up of 2D objects based on how the section plane cuts through the 3D model or a new 3D object. To create a 2D or 3D representation of the section view, you need to select the section plane object and right-click. From the right-click menu, choose Generate 2D/3D Section to display the Generate Section/Elevation dialog box (see Figure 10-3). The Generate Section/Elevation dialog box allows you to specify the type of output, the geometry in the drawing, and the destination of the output. After you've specified the desired settings, click the Create button and follow any additional prompts to insert the new block if needed.

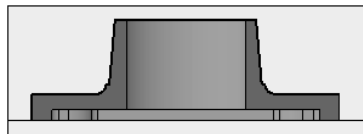


If you want additional control over how the section view looks, click the Section Settings button at the bottom of the Generate Section/Elevation dialog box to display the Section Settings dialog box (see Figure 10-4). From the Section Settings dialog box, you can specify the layer, color, linetype, and other settings used to generate the 2D or 3D section view. The dialog box has several different categories of properties that you

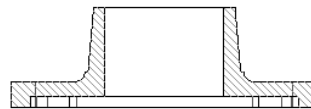
can modify, which range from how cut-away geometry might look to the fills used to represent where the section plane cuts through the 3D model. Figure 10-5 shows a 2D section created from a 3D model.



**Figure 10-4:** The Section Settings dialog box.



3D solid with live sectioning



2D section view

**Figure 10-5:** A 2D section view created from a 3D model.



1. The following exercises show how to create a section plane and use live sectioning on a 3D solid model. What steps do you take to create a 3D solid model?

1. \_\_\_\_\_
2. \_\_\_\_\_
3. \_\_\_\_\_
4. \_\_\_\_\_
5. \_\_\_\_\_
6. \_\_\_\_\_
7. \_\_\_\_\_

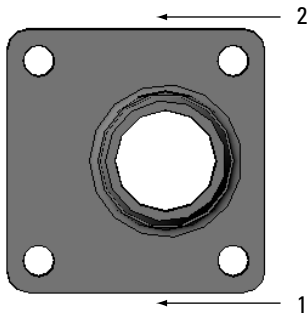


### Exercise 10-1: Creating a Section Plane and Using Live Sectioning

For this exercise, open the drawing file `Exercise10-1.dwg` from the Chapter 10 folder in the My Practice Drawings folder under My Documents. Then practice creating a section plane and activating live sectioning.

1. Choose `Draw` → `Modeling` → `Section Plane`.
2. At the `Select face or any point to locate section line or [Draw section/Orthographic] : prompt`, pick in the drawing window near the location indicated by 1 in Figure 10-6.

When selecting the location, make sure not to select a point on an object.



**Figure 10-6:**  
Creating a  
linear sec-  
tion plane.

3. At the `Specify through point : prompt`, pick in the drawing window near the location indicated by 2.
4. Select the section plane and right-click. From the right-click menu, choose `Activate Live Sectioning`.  
The order that you pick points in for the section plane determines which side are used for the section view.
5. With the section plane still selected, click the arrow-shaped grip.  
The section view flips.
6. Choose `View` → `3D View` → `SW Isometric`.  
A section view of the 3D model appears.
7. Save the drawing with the name `Completed10-1.dwg` to the Chapter 10 folder in the My Practice Drawings folder under My Documents and close the drawing.



### Exercise 10-2: Creating a 2D Section View

For this exercise, open the drawing file `Exercise10-2.dwg` or `Completed10-1.dwg` from the Chapter 10 folder in the My Practice Drawings folder under My Documents. Then practice creating a 2D section from a section plane.

1. Select the section plane object and right-click. From the right-click menu, choose `Generate 2D/3D Section`.
2. In the `Generate Section/Elevation` dialog box, make sure the following are selected:
  - `2D Section / Elevation`
  - `Include All Objects`
  - `Insert as New Block`

If you need to, click the Show Details button in the lower-left corner of the dialog box to expand it.

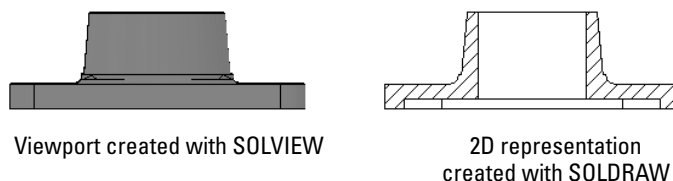
3. Click the Section Settings button.
4. In the Section Settings dialog box, make the following selections:
  - Select the 2D Section / Elevation Block Creation Settings option.
  - Under the Intersection Fill category, click Face Hatch. Select Hatch Pattern Type from its drop-down list. In the Hatch Pattern Type dialog box, select ANSI31 from the Pattern drop-down list. Click OK to return to the Section Settings dialog box.
  - Under the Intersection Fill category, click Hatch Scale and enter **0.5**.
  - Under the Cut-Away Geometry and Curve Tangency Lines categories, click Show and select No from the drop-down list.
  - Under the Background Lines and Cut-Away Geometry categories, click Hidden Line and select No from the drop-down list.
5. Click OK to return to the Generate Section/Elevation dialog box.
6. Click Create.
7. Pick a point in the drawing window and press Enter three times to insert the block.
8. Save the drawing with the name `Completed10-2.dwg` to the Chapter 10 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Viewing, drawing, and profiling solids

The SOLVIEW command creates floating viewports based on projected views of 3D solids. (I cover layouts and floating viewports in Chapter 20.) You can create floating viewports that display 3D solids in orthographic, auxiliary, and section views. After a viewport is created you can use the SOLDRAW command to convert the 3D solids in the viewport to 2D representations of the current view in the floating viewport. The SOLDRAW command is limited to floating viewports on layout tabs that are created with the SOLVIEW command.

When using the SOLDRAW command, AutoCAD creates some new layers on the fly that place hidden and visible lines and hatch objects. The layers that are created are prefixed with the name of the view that you specify when creating the floating viewport with the SOLVIEW command, and they're suffixed with `-HID`, `-VIS`, or `-HAT`. If the floating viewport represents a section view, the SOLDRAW command hatches any of the geometry that was cut through to create the view. The scale, angle, and hatch pattern name for the hatch object are based on the system variables `HPSCALE`, `HPANG`, and `HPNAME`. Figure 10-7 shows a floating viewport created with the SOLVIEW command, and then a 2D representation was created using the SOLDRAW command.

**Figure 10-7:**  
A 2D section view created with SOLVIEW and SOLDRAW.





2. How can you activate a layout tab?
- Click one of the layout tabs at the bottom of the drawing window.
  - Press Ctrl+Page Up or Ctrl+Page Down to switch between the layout tabs and Model tab in the drawing.
  - Click the Layout or Additional Layouts button on the status bar.
  - Enter **model** at the command prompt.

The SOLPROF command creates profiles of 3D solids in a floating viewport. It creates layers on the fly like the SOLDRAW command does, but instead of using the name of the view for part of the naming convention of the layers that are created, the names of the layers are based on the handle of the viewport that the profiling of the 3D solids is being done in. The lines that represent visible edges are placed on layers that are prefixed with PV-, and hidden lines are placed on layers that are prefixed with PH-. The SOLPROF command is similar to the FLATSHOT command, which I cover later in this chapter, but you can use the FLATSHOT command in Model space.

To start the SOLVIEW, SOLDRAW, and SOLPROF commands, choose Draw⇨Modeling⇨Setup and click the appropriate option or enter **solview**, **soldraw**, or **solprof** at the command prompt.

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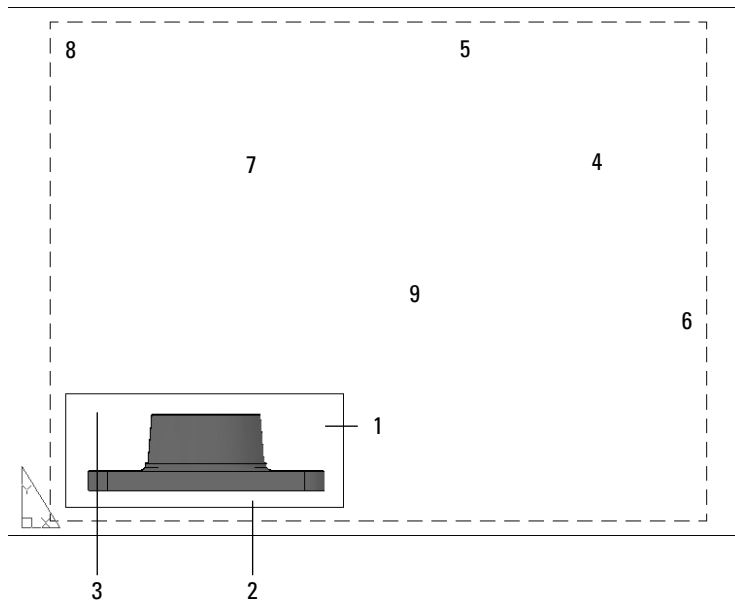
### Exercise 10-3: Creating Ortho, Auxiliary, and Section Views of a 3D Model — Part 1



For this exercise, open the drawing file `Exercise10-3.dwg` from the Chapter 10 folder in the My Practice Drawings folder under My Documents. Then practice creating different views from a 3D model on a layout tab with the SOLVIEW command.

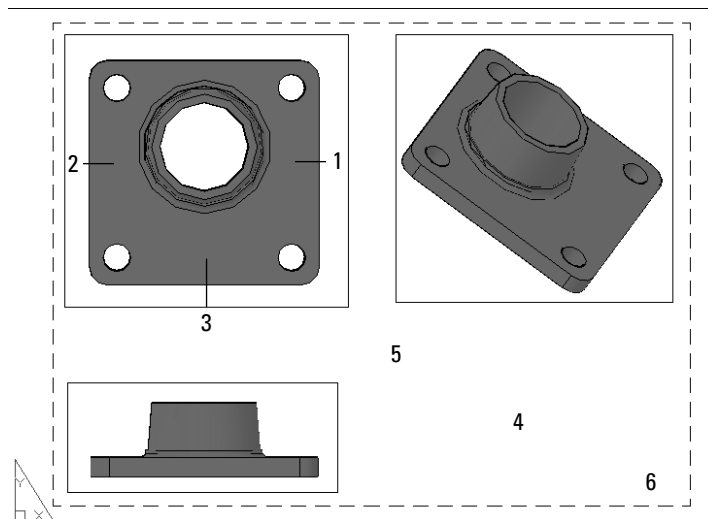
- Display the Layer Properties Manager and view the layers in the drawing. The drawing has only a few layers.
- Choose Draw⇨Modeling⇨Setup⇨View to start the SOLVIEW command.
- Right-click and choose Auxiliary.
- At the `Specify first point of inclined plane: prompt`, pick the point indicated by 1 in Figure 10-8.
- At the `Specify second point of inclined plane: prompt`, pick the point indicated by 2.
- At the `Specify side to view from: prompt`, pick the point indicated by 3.
- At the `Specify view center: prompt`, pick the point indicated by 4.
- Press Enter and pick the points near 5 and 6 to create a floating viewport.
- At the `Enter view name: prompt`, type **aux** and press Enter twice.
- Double-click the new floating viewport that you just created to enter Model space and choose View⇨Zoom⇨Extents.
- Start the SOLVIEW command again and select the Ortho option.
- At the `Specify side of viewport to project: prompt`, pick the top edge of the viewport in the lower-left corner.
- At the `Specify view center: prompt`, drag up and pick near the point indicated by 7.
- Press Enter and pick points near 8 and 9 around the view to create a floating viewport.

**Figure 10-8:**  
Layout floating viewports for orthographic and auxiliary views of a model



15. At the `Enter view name:` prompt, type **top** and press Enter once.
16. Right-click and choose Section.
17. At the `Specify first point of cutting plane:` prompt, pick the point near 1 in Figure 10-9.

**Figure 10-9:**  
Lay out a floating viewport for a section view of a model.



18. At the `Specify second point of cutting plane:` prompt, pick the point near 2.  
Make sure ORTHO is turned on.
19. At the `Specify side to view from:` prompt, pick the point indicated by 3.
20. At the `Enter view scale <1>:` prompt, press Enter.

21. At the `Specify view center:` prompt, pick the point indicated by 4.  
Make sure that ORTHO is turned off.
22. Press Enter and pick the points near 5 and 6 to create a floating viewport.
23. At the `Enter view name:` prompt, type **sec** and press Enter twice.
24. Double-click the new floating viewport that you just created to enter Model space and choose `View↔Zoom↔Extents`.
25. Double-click outside the floating viewport in the gray area in the drawing window. Exit the viewport and return to Paper space.
26. Display the Layer Properties Manager and view the layers in the drawing again. Now there are a number of new layers that AutoCAD created in the drawing as the floating viewports were being created.

---

### Exercise 10-4: Creating Ortho, Auxiliary, and Section Views of a 3D Model — Part 2



For this exercise, open the drawing file `Completed10-3.dwg` or `Exercise10-4.dwg` from the Chapter 10 folder in the My Practice Drawings folder under My Documents. Then practice creating 2D views using the `SOLDRAW` and `SOLPROF` commands from the floating viewports created with the `SOLVIEW` command.

1. At the command prompt, enter **hpname** and press Enter.
2. At the `Enter new value for HPNAME <"ANGLE">:` prompt, enter **ansi31** and press Enter.
3. Choose `Draw↔Modeling↔Setup↔Drawing` to start the `SOLDRAW` command.
4. Select the floating viewport located in the lower-right corner and press Enter to end selecting objects.

The 3D object in the floating viewport is replaced with a 2D representation of the section view.

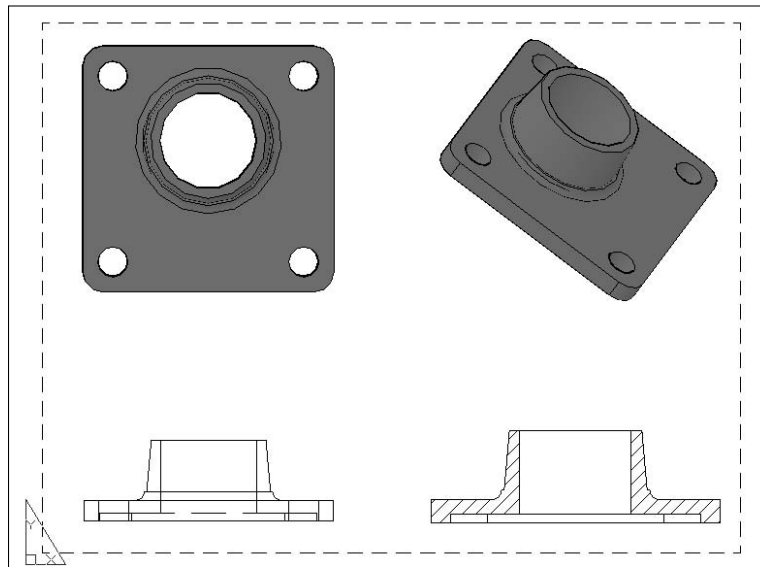
5. Double-click inside the floating viewport in the lower-left corner of the layout tab.
6. Choose `Draw↔Modeling↔Setup↔Profile` to start the `SOLPROF` command.
7. Select the 3D solid in the viewport and press Enter four times to finish the command.

By pressing Enter four times, you accept the default options of how hidden and profile lines are handled along with tangential edges.

8. Display the Layer Properties Manager and view the layers in the drawing again. Some additional lines have been created again because you used the `SOLDRAW` and `SOLPROF` commands.
9. In the Layer Properties Manager or the Layer control on the Layers toolbar (or Layers control panel on the Dashboard), click the freeze/thaw icon in the VP Freeze column for the Object layer to freeze the layer. Click OK or close the layer.

You see a 2D representation of the 3D solid in the lower-left floating viewport. Figure 10-10 shows the completed layout and all the different views that have been created.

10. Save the drawing with the name `Completed10-4.dwg` to the Chapter 10 folder in the My Practice Drawings folder under My Documents and close the drawing.



**Figure 10-10:** Layout with views of a model created with SOLVIEW, SOLDRAW, and SOLPROF.

## Flattening a Model

You spend a lot of time working on a 3D model to make it accurate, so you want to get as much use out of it as possible. One of the best uses for a 3D model is to create 2D representations of different views to share with vendors. I can think of a number of reasons that you might want to take a 3D model and generate 2D objects from them. For instance, 2D drawings help you protect your intellectual property, they're used on the shop floor for manufacturing parts, and you can use them to create exploded views of a model that can be used in an assembly guide created in a program like Corel Draw.

The FLATSHOT command allows you to create a flattened 2D representation of a 3D solid known as a *profile*. A profile of a 3D solid shows a 2D representation of a 3D solid by showing both visible and hidden edges as lines. (The FLATSHOT command is similar to the SOLPROF command, which I cover earlier in this chapter.) Unlike most commands in AutoCAD, the FLATSHOT command doesn't prompt you to select any objects from the drawing and uses all the 3D solids in Model space.

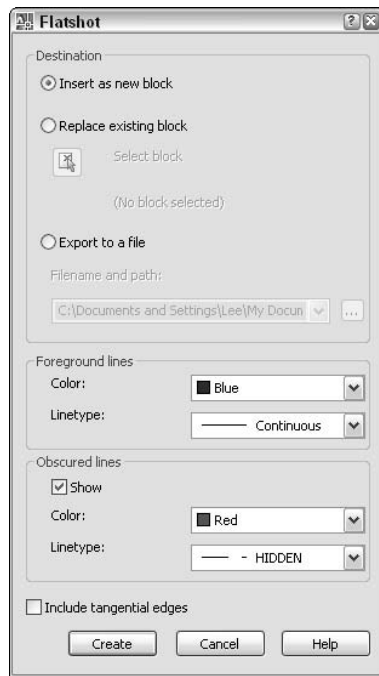
To start the FLATSHOT command, click Flatshot on the 3D Make control panel on the Dashboard or enter **flatshot** at the command prompt. The Flatshot dialog box (see Figure 10-11) opens. In the Flatshot dialog box, you can determine how the flattened view of the 3D solids is created; you can create a block in the current drawing or create a new drawing file of the flattened view.

You can specify the color and linetypes that are used for both the foreground and obscured lines. You would see the *foreground lines* when viewing the 3D solid with the 3D Hidden visual style, and the *obscured lines* are typically hidden. Examples of obscure lines might be from an inset, holes through the 3D solid, or any other back lines that might be hidden otherwise from the current viewing angle. When a flattened view is created with the FLATSHOT command, no new layers are automatically created on the fly like they are with the SOLPROF command for both visible and hidden lines. Figure 10-12 shows a 2D representation of a flattened view of a 3D solid from the top.

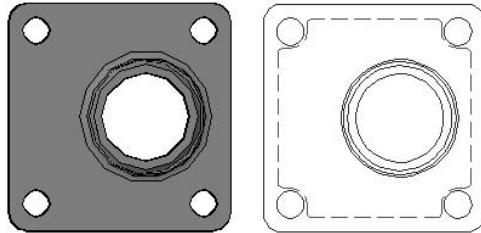


The block that gets inserted is placed on the current UCS. If you want the block to be inserted so it appears in the current view, you want to set the UCS to match the view. I cover working with the UCS in Chapter 5.





**Figure 10-11:** The Flatshot dialog box allows you to steamroll a model.



**Figure 10-12:** The top view of a 3D solid flattened.



AutoCAD ships with a set of additional utilities called Express Tools. One of these tools is called Flatten, and it does exactly what its name implies. The `FLATTEN` command (Express→Modify→Flatten Objects) converts 3D objects to 2D objects. Unlike with the `FLATSHOT` command, you can select which objects you want to flatten but the objects you select are removed from the drawing at the same time. To find out more about the `FLATTEN` command, choose Express→Help to open the online help system that comes with the Express Tools.



### Exercise 10-5: Flattening a 3D Solid

For this exercise, open the drawing file `Exercise10-5.dwg` from the Chapter 10 folder in the My Practice Drawings folder under My Documents. Then practice creating a flattened view of a 3D model.

1. Choose View→3D View→Top.
2. At the command prompt, type **flatshot** and press Enter.
3. In the Flatshot dialog box, select Insert as New Block under the Destination section if it isn't already selected.
4. Under the Foreground Lines section, select Blue from the Color drop-down list.
5. Under the Obscured Lines section, select the Show check box if it isn't selected and select Red from the Color drop-down list.

6. Select Other from the Linetype drop-down list under the Obscured Lines section to load a linetype.
7. In the Select Linetype dialog box, click Load.
8. In the Load or Reload Linetypes dialog box, select HIDDEN from the Available Linetypes list and click OK to return to the Select Linetype dialog box.
9. Select HIDDEN in the Select Linetype dialog box and click OK to return to the Flatshot dialog box.
10. Click Create.
11. Click in the drawing window to insert the block and press Enter three times to finish inserting the view.
12. Save the drawing with the name `Completed10-5.dwg` to the Chapter 10 folder in the My Practice Drawings folder under My Documents and close the drawing.

## *Answers to Practice Opportunities*

### **1. What steps do you take to create a 3D solid model?**

1. Start a new drawing using the `acad3d.dwt` drawing template.
2. Set the drafting settings that you want to use.
3. Define any coordinate systems that you might need to create the 3D solids for your model.
4. In the drawing, add the 3D solids that you want to define your 3D model with.
5. Modify the 3D solids as necessary.
6. Create any named views that you want to use when working in the drawing.
7. Use visual styles as needed to view your 3D model.

See Chapter 8 for more information.

### **2. How can you activate a layout tab?**

A, B, C. You can click one of the layout tabs, press `Ctrl+Page Up` or `Ctrl+Page Down` to switch between tabs, or click the `Layout` or `Additional Layouts` buttons on the status bar. You can't enter the `MODEL` command to manipulate the layout tabs.

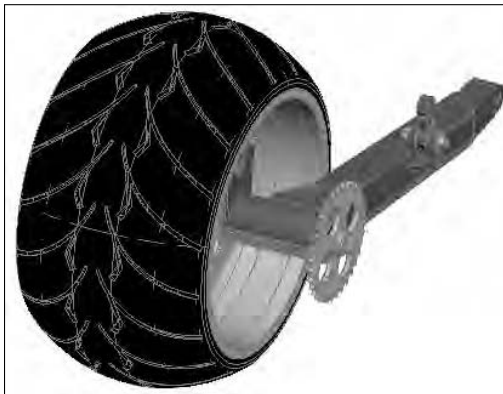
## Chapter 11

# Additional Exercises: Modeling with Solids

### *In This Chapter*

- ▶ Reinforcing concepts by creating models with 3D solids
- ▶ Reinforcing concepts used for modifying 3D solids

The previous chapters of Part III tell you about creating and modifying 3D solids to create 3D models in AutoCAD. The 3D solids that you create range from primitives in the forms of boxes, spheres, cones, and others. In those chapters, you also create complex 3D solids using techniques such as lofting and sweeping. The exercises in this chapter are a continuation of the exercises in Chapter 7. Figure 11-1 shows the rear assembly of the custom motorcycle that you work with in this chapter.



**Figure 11-1:**  
The rear  
assembly of  
a custom  
motorcycle.

Model courtesy of Mark Hechel of MasterGraphics, Inc.

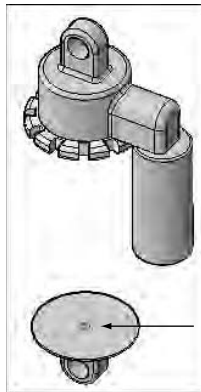
## *Modeling with Primitive and Complex 3D Solids*

In this series of exercises, you finish a 3D part that you link to the rear assembly of the custom motorcycle. The 3D part is a shock that's a purchased part and not a part that will be manufactured. When working with purchased parts in your models, you often don't start with 2D drawings that you create because you won't be manufacturing the parts yourself. In these cases, you must obtain a physical part and take measurements from it to create a 3D part for use in your models.

### Exercise 11-1: Creating 3D Primitives of the Solid Kind

For this exercise, open the drawing file `Exercise11-1.dwg` from the Chapter 11 folder in the My Practice Drawings folder under My Documents. Then practice creating primitive 3D solids to finish part of the 3D model of the shock.

1. Set the layer Shock current.
2. Choose Draw→Modeling→Cone.
3. When prompted to specify the center point of the base for the cone, use the Node object snap and select the point indicated by the arrow in Figure 11-2.



**Figure 11-2:**  
A partially  
finished 3D  
model of the  
shock.

*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

4. Enter **1.2** for the base radius and press Enter.
5. At the Specify height or [2Point/Axis endpoint/Top radius] <15.0949>: prompt, enter **T** and press Enter.
6. Enter **0.75** for the top radius and press Enter.
7. Enter **1.27** for the height and press Enter.
8. Select the cone that you just created and set its color to 45.

Changing the color of an object directly usually isn't a good drafting practice, but it's an effective way to add color to 3D solids for visualization purposes without making the layering scheme overly complex.



To set the cone's color, select the cone and right-click. From the right-click menu, choose Properties. On the Properties palette, choose Select Color from the Color property's drop-down list to display the Select Color dialog box. In the Select Color dialog box, click the Index Color tab and select the color 45 swatch or enter **45** in the Color text box. When done, click OK.

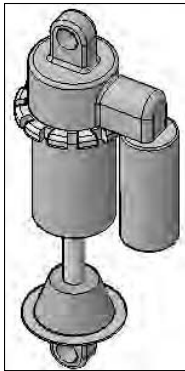
9. Choose Draw→Modeling→Cylinder.
10. When prompted to specify the center point of the base for the cylinder, use the Center object snap and select the center point at the top of the cone that you just created.
11. Enter **0.31** for the base radius and press Enter.
12. Enter **2.56** for the height and press Enter.
13. Select the cylinder that you just created and set its color to 219,219,219 using the True Color tab of the Select Color dialog box.



To set the cylinder's color, select the cylinder and right-click. From the right-click menu, choose Properties. On the Properties palette, choose Select Color from the Color property's drop-down list to display the Select Color dialog box. In the Select Color dialog box, click the True Color tab and select 219,219,219 in the color palette. Or simply type the value in the Color text box at the bottom. When you're done, click OK.

14. Start the CYLINDER command again.
15. When prompted to specify the center point of the base for the cylinder, use the Center object snap and select the top of the cylinder that you just created.
16. Enter **1.25** for the base radius and press Enter.
17. Enter **3.78** for the height and press Enter.

Now the model looks like the one in Figure 11-3. However, it isn't quite done yet — see the following exercise.



**Figure 11-3:**  
A nearly completed 3D model of the shock.

*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

18. Save the drawing with the name `Completed11-1.dwg` to the Chapter 11 folder in the My Practice Drawings folder under My Documents and close the drawing.

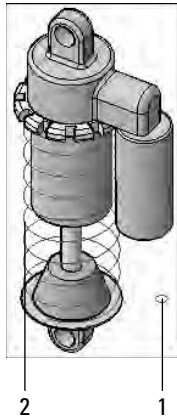
### Exercise 11-2: Sweeping a 2D Profile to Create a Complex 3D Solid



In this exercise, you practice creating the spring for the shock through sweeping a circle for the diameter of the spring along a 2D path. The 2D path in this exercise is a 3D polyline instead of a helix because it needed to be compressed at both the top and bottom of the spring.

1. Open the drawing `Exercise11-2a.dwg` from the Chapter 11 folder in the My Practice Drawings folder under My Documents.
2. Turn on the layer `3DConst_spring`.  
The drawing looks like the one in Figure 11-4, with a path and a circle shown.
3. Set the layer `Shock_spring` current.
4. Choose `Draw` ⇨ `Modeling` ⇨ `Sweep`.
5. When prompted for the object to sweep, select the circle indicated by 1 in Figure 11-4 and press Enter.
6. When prompted for the path, select the 3D polyline indicated by 2.
7. To get a better look at the new 3D solid that you just created, choose `View` ⇨ `Visual Styles` ⇨ `Visual Styles Manager` to display the Visual Styles Manager palette.

**Figure 11-4:**  
A path for sweeping a circle to create the 3D spring.

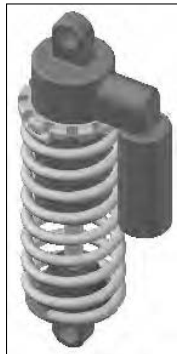


Model courtesy of Mark Hechel of MasterGraphics, Inc.

8. On the Visual Styles Manager palette, double-click Realistic under the Available Visual Styles in Drawing section to set the visual style current.
9. Under the Edge Settings category, select Isolines from the Edge Mode property's drop-down list.

All the isolines are removed from the drawing, giving it a much cleaner look. The drawing looks similar to the one in Figure 11-5.

**Figure 11-5:**  
Giving the shock some style.



Model courtesy of Mark Hechel of MasterGraphics, Inc.

10. Save the drawing with the name `Complete11-2a.dwg` to the Chapter 11 folder in the My Practice Drawings folder under My Documents and close the drawing.
11. Open the drawing `Exercise11-2b.dwg` from the Chapter 11 folder in the My Practice Drawings folder under My Documents.
12. Choose `Insert > DWG Reference` to display the Select Reference File dialog box.
13. Browse to the Chapter 11 folder in the My Practice Drawings folder under My Documents and select `Complete11-2a.dwg`.
14. In the External Reference dialog box, select the following options:
  - Select Attachment under the Reference Type section.
  - Set Path Type to Relative Path.
  - Select Specify On-Screen under the Insertion Point section.
  - Deselect the other Specify On-Screen check boxes under the Scale and Rotation sections.

15. Click OK.
16. For the insertion point of the drawing reference (xref), pick the point object indicated by the arrow in Figure 11-6 using the Node object snap.

**Figure 11-6:**  
Referencing  
in the shock  
3D model.



*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

17. Choose Modify→3D Operations→3D Rotate.
18. When prompted to select the objects to rotate, select the 3D model of the shock and press Enter.
19. For the base point of the rotation operation, pick the point object that you selected to attach the 3D model of the shock using the Node object snap.
20. Click over the red ribbon of the rotation gizmo to constrain the rotation of the shock along the X axis.
21. When prompted for the angle of rotation, enter **-52** and press Enter.
22. Turn on the layers Shock\_bushings and Shock\_fasteners.

The model now looks like the one in Figure 11-7. It's finally complete.

**Figure 11-7:**  
A finished  
model after  
the shock is  
rotated into  
position.



*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

23. Save the drawing as the name `Completed11-2b.dwg` to the Chapter 11 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Modifying a 3D Solid

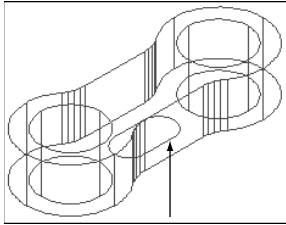
In this series of exercises, you modify the 3D part (the transfer link) that you created in Exercise 8-3. You create insets on the part and add some fillets to finish the design of the part. After modifying the 3D model of the transfer link, you open the rear assembly of the custom motorcycle to see the changes live in the model assembly. Because the transfer link model was originally referenced and not inserted into the drawing with the rear assembly, the changes show up automatically with minimal work.

### Exercise 11-3: Subtracting from a 3D Solid

For this exercise, open the drawing file `Exercise11-3.dwg` from the Chapter 11 folder in the My Practice Drawings folder under My Documents. Then practice extruding and subtracting 3D solids to create insets in the 3D model of the transfer link.

1. Choose Draw→Modeling→Extrude.
2. When prompted to select objects, select the small region located at the bottom of the 3D solid and press Enter to end selecting objects. The region is indicated by the arrow in Figure 11-8.

**Figure 11-8:**  
Creating insets using the EXTRUDE and SUBTRACT commands.

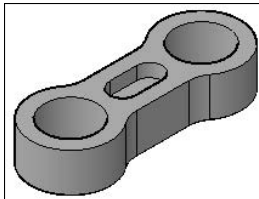


*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

3. When prompted for the height, drag the crosshairs upward in the drawing window. Then enter **0.25** and press Enter.
4. Choose Modify→Copy.
5. When prompted for the base point, enter **0,0,0.5** and press Enter twice.
6. Choose Modify→Solid Editing→Subtract.
7. When prompted for the 3D solids to subtract from, select the large 3D solid and press Enter.
8. When prompted for the 3D solids to subtract, select the two 3D solids that you recently created and press Enter.
9. Switch to the Conceptual visual style so you can see the new insets.

Figure 11-9 shows what the model looks with the visual style applied.

**Figure 11-9:**  
Insets added to the transfer link.



*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

10. Save the drawing with the name `Completed11-3.dwg` to the Chapter 11 folder in the My Practice Drawings folder under My Documents and close the drawing.

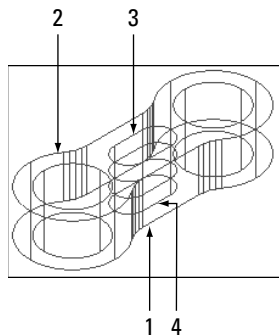




### Exercise 11-4: Adding Fillets to a 3D Solid

In this exercise, you practice adding fillets to the 3D model of the transfer link to complete the part based on the original 2D drawing from Exercise 8-3.

1. Open the drawing `RearAssembly.dwg` from the Chapter 11 folder in the My Practice Drawings folder under My Documents.  
The transfer link doesn't contain any of the insets or fillets that you created in Exercise 11-3. The reason for this is that the file `Transfer_link.dwg` is linked into the drawing and not `Exercise11-3.dwg`. You update the `Transfer_link.dwg` file near the end of this exercise to see the changes to the part in the assembly.
2. Close the drawing `RearAssembly.dwg` without saving any changes made to it.
3. Open the drawing `Exercise11-4.dwg` from the Chapter 11 folder in the My Practice Drawings folder under My Documents.
4. Choose `Modify`→`Fillet`.
5. When prompted to select the first object, pick the edge indicated by 1 in Figure 11-10.

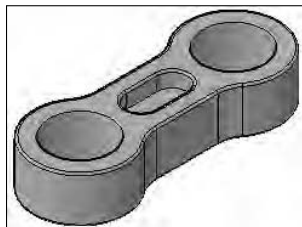


**Figure 11-10:**  
Adding fillets.

*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

6. When prompted for the fillet radius, enter **0.03** and press Enter.
7. When prompted to select an edge, enter **C** and press Enter to start the chain option.
8. When prompted to select an edge, select the same edge that you selected previously and continue selecting the edges indicated by 2, 3, and 4. Press Enter to finish selecting edges to fillet.
9. Switch to the Conceptual visual style so you can see the new fillets. Figure 11-11 shows how the model looks with the visual style applied.

**Figure 11-11:**  
A completed transfer link with filleted edges.



*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

- 10.** Save the drawing with the name `Transfer_link.dwg` to the Chapter 11 folder in the My Practice Drawings folder under My Documents.

This is the name of the file that's attached to the `RearAssembly.dwg` file; normally, you would just work with the file that is attached to the drawing in your assembly.

- 11.** Save the drawing again, but this time with the name `Completed11-4.dwg` to the Chapter 11 folder in the My Practice Drawings folder under My Documents. Then close the drawing.

- 12.** Reopen the `RearAssembly.dwg` file.

Both the insets that were added in Exercise 11-3 and the fillets that you just added are now displayed on the transfer link within the assembly.

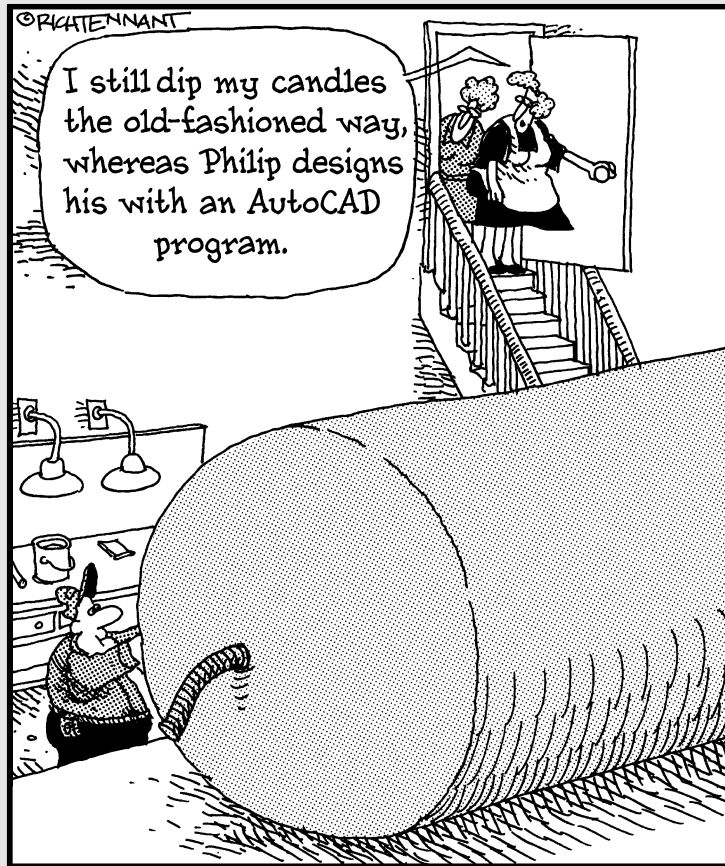
- 13.** Close the `RearAssembly.dwg` file without saving any changes made to it.

# Part IV

## 3D Modeling — Surfaces

### The 5<sup>th</sup> Wave

By Rich Tennant



### *In this part . . .*

**3**D solids aren't the only objects that you can use to create 3D models in AutoCAD. AutoCAD offers the ability to create surface models using faces, meshes, and surface objects. Based on the object you're modeling, you likely use a combination of faces, meshes, and surface objects. Surface models aren't as accurate as 3D solids because they're created from objects that don't support true curved objects. Instead curved edges on a surface model are created by controlling the density of the straight edges of a mesh to give the model a smooth look.

## Chapter 12

# Modeling with Surfaces

### *In This Chapter*

- ▶ Working with surfaces
- ▶ Creating 3D faces
- ▶ Creating 3D mesh primitives
- ▶ Creating planar surfaces and surface objects

**I**n this chapter, you take a look at surface modeling, which is much different from modeling with solids. Surface models don't offer as much flexibility in the area of editing as 3D solids do, but in some situations, you'll need to use surfaces. For example, you need a surface model for a plane below a 3D model to receive shadows that are cast for added realism, or it can be an open object that doesn't need any thickness. The exercises in this chapter show you how to create 3D faces, 3D mesh primitives, and planar surfaces.

## *Working with Surfaces*

You can use surfaces in place of 3D solids when creating 3D models, but they take some upfront planning to make sure that you don't spend a lot of time editing them. Like 3D solids, surfaces create a virtual representation of an object that you plan on manufacturing or building in the physical world.

Surface models in AutoCAD are made up of three different object types: faces, meshes, and surface objects:

- ✓ **Faces** are the most basic object type that you can create a surface model from. A face is made up of either three or four sides and can hide objects beyond the face.
- ✓ A **mesh** is simply a combination of multiple faces that are arranged to create a single object that can represent an object like a box or sphere.
- ✓ A **surface object** can be flat or created through an operation such as extruding or lofting. Surface objects are much more complex in nature than a mesh is and aren't just made up of a series of points. AutoCAD allows you to create surface models through primitives and complex objects just like 3D solids.

Because meshes are created from a series of faces and not curved elements, they aren't nearly as visually accurate as a 3D solid is. So why use curved mesh objects if they aren't really curved objects? Over a decade ago, computers weren't as powerful as they are today, and it took a lot of computing power and storage space to use 3D solids. So companies resorted to using surface models to help their clients visualize what the end product would look like. 2D drawings were the main way to communicate designs and develop products, but times have changed. These days, storage and processing power are relatively cheap, which allows for the use of 3D solids for creating conceptual designs and accurate models.

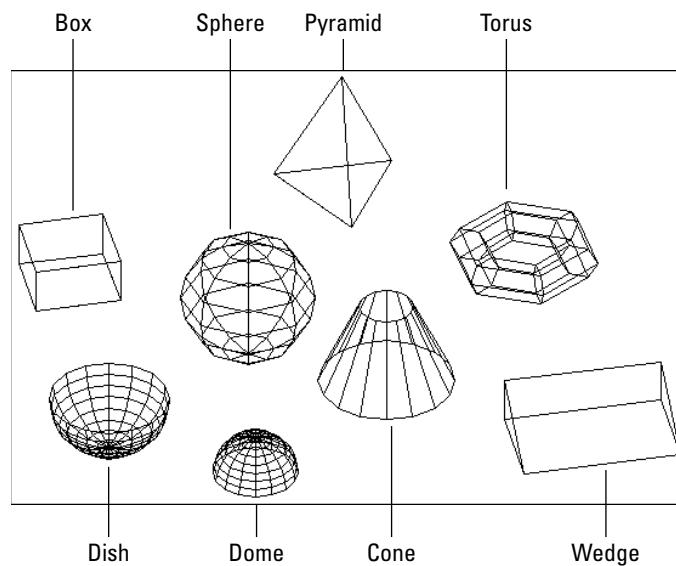


Other software packages you might work with might not handle 3D solids very well, so you might have to use surfaces instead. So it really depends on what you can use in your drafting environment; if the model needs to be lightweight and portable, you might have to use surfaces.

Autodesk is placing more emphasis on using 3D solids over surfaces because of the number of enhancements to 3D solids in recent releases. In fact, most of the surface creation commands have been removed from the menus, toolbars, or dashboard panels. To create 3D faces and mesh objects, you can use a few of the remaining commands that have fortified themselves on the Draw menu. But most of the commands to create 3D mesh objects are now accessible only from the command prompt.

## Primitive surfaces

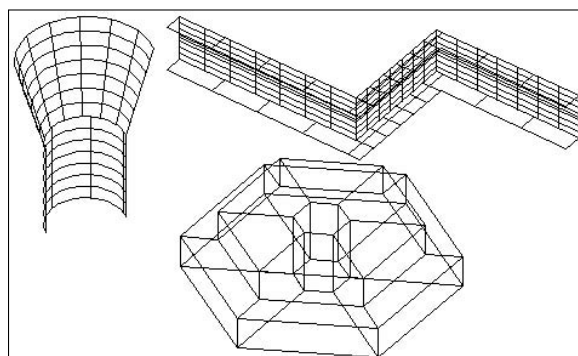
*Primitive surfaces* are 3D meshes created in the form of basic geometric shapes, and they're often used as a starting point for creating a surface model. AutoCAD allows you to create eight different 3D mesh primitives (see Figure 12-1). Although 3D mesh primitives aren't quite as defined as their 3D solid counterparts are, they're categorized into two distinct groups: straight edges and curves. But 3D meshes really aren't curved objects: Like in an optical illusion, the "curved" edges are really made of straight segments that appear curved from a distance.



**Figure 12-1:**  
3D mesh  
primitives.

## Complex surfaces

AutoCAD allows you to create complex objects made up of either meshes or surface objects from open or closed 2D objects (also known as *profiles*) through operations such as revolving, lofting, or extruding (see Figure 12-2). Some of the same commands that create complex 3D solids also create complex surface objects. I cover creating both complex 3D meshes and surface objects in Chapter 13.

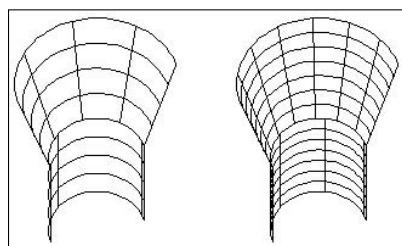


**Figure 12-2:**  
Complex 3D  
meshes and  
surface  
objects.

## *Controlling the smoothness of a surface*

The current values of FACETRES and VIEWRES affect how models created with 3D solids appear in the drawing window. How surface models are displayed in the drawing window is somewhat more complicated based on the object types. 3D faces and meshes always look the same no matter what the current values of FACETRES and VIEWRES are, but the same doesn't hold true for surface objects. Surface objects are affected by the current value of VIEWRES just like arcs, circles, and curved faces on 3D solids. A low zoom percentage for VIEWRES causes curved surface objects to appear blocky.

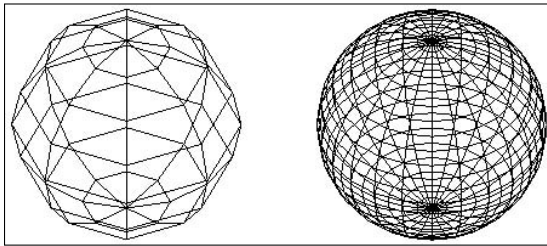
The system variables SURFU and SURFV affect the number of isolines that appear on new surface objects that you create for your models. You can change the SURFU and SURFV system variables by entering the variable name at the command prompt, or using the sliders for the U and V isolines under the 3D Objects section on the 3D Modeling tab of the Options dialog box. Figure 12-3 shows two surface objects created through lofting with different U and V isoline values.



**Figure 12-3:**  
Lofted sur-  
face object  
with differ-  
ent U and  
V isoline  
values.

Another factor that affects how smooth surface models appear is the density in which you create 3D meshes. 3D meshes — unlike surface objects — are made of straight segments, so when creating the mesh, you need to know the number of segments that you want. Unfortunately, if the mesh isn't smooth enough, you need to re-create it. Some of the commands that allow you to create 3D meshes prompt you for the number of segments, but some don't. If you aren't prompted for the number of segments, use the system variables SURFTAB1 and SURFTAB2. The system variables that you need to change are based on the command that you're using to create the mesh. Figure 12-4 shows two 3D mesh spheres created with two different segment counts.

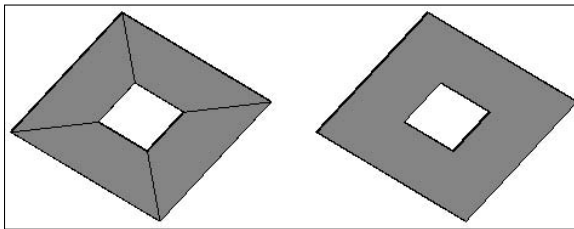
**Figure 12-4:**  
3D mesh  
spheres  
with differ-  
ent segment  
counts.



## Putting the Face in Surface

*Faces* are elements of a 3D solid, but they're also objects that create a three- or four-sided object to make a surface model. As you model with faces to create objects, a number of faces are stitched together like a patchwork quilt. You can control which edges of a face are displayed to make it appear as if the adjacent faces create a single object. Figure 12-5 shows two rectangular objects with a rectangular hole in the center created using 3D faces. The faces on the left have all their edges visible, whereas the one on the right has select edges hidden.

**Figure 12-5:**  
Creating  
objects from  
3D faces.



You start the command to create a 3D face by choosing Draw⇨Modeling⇨Meshes⇨3D Face or entering **3dface** at the command prompt.



To view the invisible edges of a 3D face, enter **spframe** at the command prompt. When prompted for a new value, enter **1** to show the edges marked as invisible or **0** to hide the edges marked as invisible. Then you need to force a regeneration of the drawing by using the REGEN (View⇨Regen) command.



1. In the following exercise, you create a 3D face. What steps do you think you take to create a 3D face?

1. \_\_\_\_\_
2. \_\_\_\_\_
3. \_\_\_\_\_



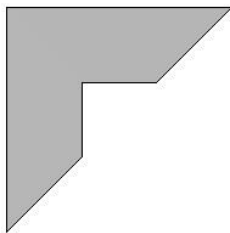


### Exercise 12-1: Creating a 3D Face

For this exercise, open the drawing file `Exercise12-1.dwg` from the Chapter 12 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D face.

1. Turn off Dynamic Input.  
To turn off Dynamic Input, click the DYN button on the status bar so it appears popped out.
2. Choose `Draw` → `Modeling` → `Meshes` → `3D Face`.
3. At the `Specify first point or [Invisible]:` prompt, enter `0,0,0` and press Enter.
4. At the `Specify second point or [Invisible]:` prompt, enter `2,2,2` and press Enter.
5. At the `Specify third point or [Invisible] <exit>:` prompt, enter `I` and press Enter to draw the edge as invisible.
6. The prompt repeats, and when prompted for the third point enter `2,4,2` and press Enter to draw the edge as invisible.
7. At the `Specify fourth point or [Invisible] <create three-sided face>:` prompt, enter `0,6,0` and press Enter twice to end the command.  
Only three lines are displayed, but if you use one of the 3D visual styles like Conceptual you can see that the face object has been drawn.
8. Start the 3DFACE command again. At the first point prompt, enter `I` and press Enter. When the prompt repeats, enter `@` and press Enter to pick the last point that created the previous 3D face object.
9. Enter `2,4,2` and press Enter for the second point.
10. Enter `4,4,2` and press Enter for the third point.
11. Enter `6,6,0` and press Enter twice to complete the face.  
The second face is created and the shared edges of the two faces are set as invisible, causing the two faces to look like they form one surface (see Figure 12-6).

**Figure 12-6:**  
Two faces  
drawn to  
look like  
a single  
surface.



12. Save the drawing with the name `Completed12-1.dwg` to the Chapter 12 folder in the My Practice Drawings folder under My Documents and close the drawing.

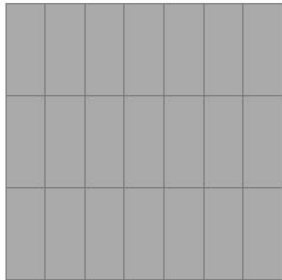
## Meshes with Straight Edges

3D mesh objects come in a variety of shapes and sizes. AutoCAD allows you to create a planar mesh object and three different 3D mesh primitives that are made up of straight edges. The 3D mesh primitives with straight edges that you can create are very similar

to their 3D solid cousins: box, wedge, and pyramid. 3D planar meshes — although not primitives — share some features in common with the 3D mesh primitives; so they're in this chapter as well.

## Planar meshes

*Planar meshes* are similar to a 3D face except that they're divided into segments that run in the M and N directions. The M and N directions are one of the strange things you have to deal with for 3D objects. The direction might not be in the X or Y directions, so instead of saying horizontal and vertical, you use M and N. The M direction creates horizontal segments, and the N direction creates vertical segments. Figure 12-7 shows a 3D planar mesh object created with a mesh size of 4 in the M direction (going from up to down) and 8 in the N direction (going from left to right).



**Figure 12-7:**  
A planar  
mesh (not  
mess).

You can start creating a 3D mesh by entering **3d** at the command prompt and choosing the Mesh option or entering **ai\_mesh** at the command prompt. When you've created the 3D mesh object, you can use grip editing to manipulate the flow of the mesh.

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### Exercise 12-2: Creating a 3D Planar Mesh

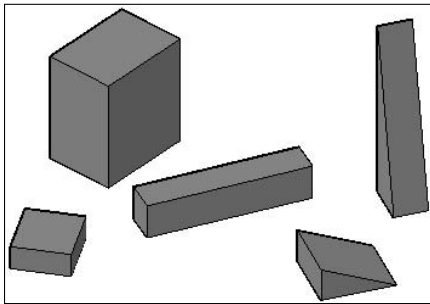
For this exercise, open the drawing file `Exercise12-2.dwg` from the Chapter 12 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D mesh.

1. Turn off Dynamic Input.
2. At the command prompt enter **3d** and press Enter.
3. At the Enter an option [Box/Cone/DIsh/Dome/Mesh/Pyramid/Sphere/Torus/Wedge]: prompt, enter **M** and press Enter to draw a planar mesh object.
4. At the Specify first corner of mesh: prompt, enter **0,0,0** and press Enter.
5. At the Specify second corner of mesh: prompt, enter **3,6,2** and press Enter.
6. At the Specify third corner of mesh: prompt, enter **6,6,2** and press Enter.
7. At the Specify fourth corner of mesh: prompt, enter **6,0,0** and press Enter.
8. At the Enter mesh size in the M direction: prompt, enter **4** and press Enter.
9. At the Enter mesh size in the N direction: prompt, enter **4** and press Enter.
10. Set the Conceptual visual style (or another visual style) current.

11. Switch to an isometric view using either one of the orbiting commands or options from the View→3D View sub-menu.
12. Save the drawing with the name `Completed12-2.dwg` to the Chapter 12 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Boxes and wedges

*Boxes* and *wedges* are very similar in how they are created. When you create a box or wedge, you specify its length, width, and height. As you create a box, you have options to create a cube (all sides are the same length) and the box can be rotated after you specify its length, width, and height. Figure 12-8 shows some of the different 3D mesh boxes and wedges that you can create. You might create a box to represent a pedestal to mount a flange or to create a housing for a fuse box. Wedges are useful when you want to create an edge of a blade or an inclined/sloped plane.



**Figure 12-8:**  
Boxes and  
wedges.

To start the command to create a 3D mesh primitive of a box or wedge, enter **3d** at the command prompt and choose the Box or Wedge option. Or enter at the command prompt **ai\_box** for a box or **ai\_wedge** for a wedge.



2. What option is available when creating a 3D mesh primitive of a box or wedge that is not available when creating a 3D solid primitive of a box or wedge?
  - A. You can rotate the box or wedge after the dimensions of the primitive have been specified.
  - B. There is no difference.
  - C. You can create a cube, making all edges the same length.

### Exercise 12-3: Creating a 3D Mesh Box or Wedge

For this exercise, open the drawing file `Exercise12-3.dwg` from the Chapter 12 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D mesh box.

1. At the command prompt, enter **3d** and press Enter.
2. At the Enter an option [Box/Cone/DIsh/Dome/Mesh/Pyramid/Sphere/Torus/Wedge]: prompt, enter **b** and press Enter.
3. At the Specify corner point of box: prompt, pick a point in the middle of the drawing window.
4. At the Specify length of box: prompt, enter **6** and press Enter.



You can also specify a point on-screen by picking in the drawing window. You won't see the creation of the box interactively like you do when using the BOX command to create a 3D solid of the primitive.

5. At the Specify width of box or [Cube]: prompt, enter **3** and press Enter.
6. At the Specify height of box: prompt, enter **1.5** and press Enter.

Unlike the BOX command, you can rotate the object around the corner point that you picked initially.

7. At the Specify rotation angle of box about the Z axis or [reference]: prompt, enter **0** and press Enter.

You can't just press Enter and expect the box to not be rotated; AutoCAD uses the current angle that is being shown in the drawing window to rotate the final object.

8. Set the Conceptual visual style current or another visual style of your choosing, and change the viewing angle so you're looking at the box from the side.
9. Save the drawing with the name `Completed12-3.dwg` to the Chapter 12 folder in the My Practice Drawings folder under My Documents and close the drawing.



3. Exercise 12-3 shows how to create a 3D mesh primitive of a box. What steps do you take to create a 3D mesh primitive of a wedge?

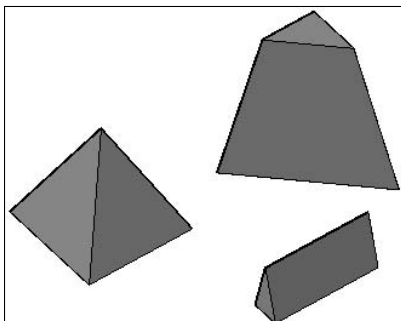
1. \_\_\_\_\_
2. \_\_\_\_\_
3. \_\_\_\_\_
4. \_\_\_\_\_

## Pyramid

*Pyramids* are traditionally objects with four equal sides, but AutoCAD bends the rules a bit and allows you to create a pyramid with four sides that aren't necessarily equal in length. Creating a pyramid also isn't limited to just four sides; you can create a three-sided geometric object known as a *tetrahedron*.

You start the 3D mesh primitive for a pyramid by entering **3d** at the command prompt and choosing the Pyramid option or entering **ai\_pyramid** at the command prompt. After you define the base of the pyramid, you can specify an apex (peak), ridge or a top that's defined with the same number of edges as the base. Figure 12-9 shows some of the different 3D mesh pyramids that you can create.

**Figure 12-9:** Three- and four-sided pyramids that can be created as 3D meshes.



### Exercise 12-4: Creating a 3D Mesh Pyramid

For this exercise, open the drawing file `Exercise12-4.dwg` from the Chapter 12 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D mesh pyramid.



1. Make sure ortho mode is turned on.  
To turn on ortho mode, click the ORTHO button on the status bar so it appears indented.
2. At the command prompt enter **3d** and press Enter.
3. At the Enter an option [Box/Cone/DIsh/Dome/Mesh/Pyramid/Sphere/Torus/Wedge]: prompt, enter **P** and press Enter to draw a pyramid.
4. At the Specify first corner point for base of pyramid: prompt, enter **0,0,0** and press Enter.
5. At the Specify second corner point for base of pyramid: prompt, drag the crosshairs to the right. Then enter **4.5** and press Enter.
6. At the Specify third corner point for base of pyramid: prompt, drag the crosshairs downward. Then enter **2.5** and press Enter.
7. At the Specify fourth corner point for base of pyramid or [Tetrahedron]: prompt, drag the crosshairs to the left. Then enter **4.5** and press Enter or enter **T** to create a three-sided pyramid.
8. At the Specify apex point of pyramid or [Ridge/Top]: prompt, enter **2.25,-1.25,4** and press Enter. Or enter **r** to create a ridge, which allows you to create an edge that's created by the ends of the pyramid coming together into two points instead of one. Or enter **T** to create the top of the pyramid with a flat or sloped face based on how the points are selected.



Before you create a pyramid, you can create point objects that you can snap to and make it easier to define the ridge or top of the pyramid. The point objects serve as reference points that you can erase later.

9. Set the Conceptual visual style (or another visual style) current and change the viewing angle so you're looking at the pyramid from the side.
10. Save the drawing with the name `Completed12-4.dwg` to the Chapter 12 folder in the My Practice Drawings folder under My Documents and close the drawing.



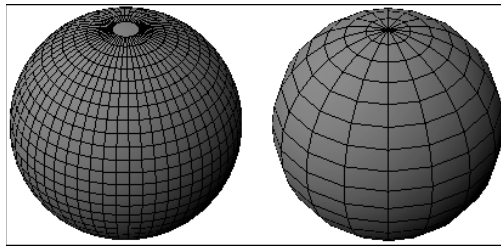
4. Exercise 12-4 shows how to create a four-sided pyramid with an apex. What steps do you take to create a three-sided pyramid (tetrahedron) with a top and not an apex?

1. \_\_\_\_\_
2. \_\_\_\_\_
3. \_\_\_\_\_
4. \_\_\_\_\_
5. \_\_\_\_\_

## Curvy Meshes

Although 3D meshes are all created from straight edges, some 3D meshes give the sense of curved edges. Curved meshes don't appear as smooth as a curved 3D solid does up close, but from a distance, you can't tell which one is which. 3D meshes aren't used to create accurate models because they look jagged; if jagged-looking edges don't work for you, don't use curved meshes — or at least don't use them when curves are involved. Figure 12-10 shows two spheres: one created as a 3D solid and the other as a 3D mesh with the Conceptual visual style current. If you look along the outer edges of the sphere on the right you can see the jagged edges. To create a smoother-looking sphere as a 3D mesh, increase its segment count.

**Figure 12-10:**  
To solid or  
to mesh,  
that is the  
question.



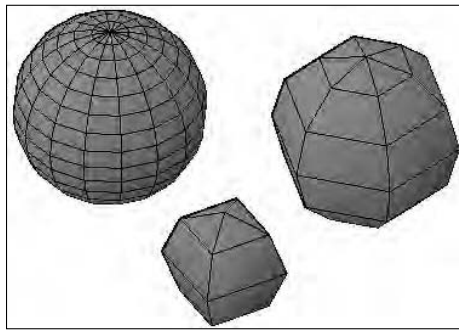
3D solid sphere

3D mesh sphere

## Sphere

*Spheres* (see Figure 12-11) are the three-dimensional counterparts to a circle, but they're defined differently because they require additional input.

**Figure 12-11:**  
Spheres,  
some of  
which don't  
even look  
like spheres.



You start the 3D mesh sphere by entering **3d** at the prompt and choosing the Sphere option or entering **ai\_sphere** at the command prompt. Then you define the sphere by specifying a center point, the radius or diameter, and the number of longitudinal and latitudinal segments to carve the sphere into. The more segments used, the smoother the 3D mesh appears. Longitudinal segments run from the top to the bottom of the sphere, and latitudinal segments run around the sphere. Spheres are useful for the ball in a mouse, the top of a screw, or a handle with a rounded end. Because you create a sphere based on its center point, half the sphere is created above and below the current working plane or the specified point.

### Exercise 12-5: Creating a 3D Mesh Sphere

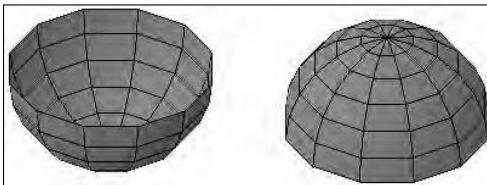
For this exercise, open the drawing file `Exercise12-5.dwg` from the Chapter 12 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D mesh sphere.

1. At the command prompt enter **3d** and press Enter.
2. At the Enter an option [Box/Cone/DIsh/Dome/Mesh/Pyramid/Sphere/Torus/Wedge]: prompt, enter **S** and press Enter to draw a sphere.
3. At the Specify center point of sphere: prompt, pick a point near the middle of the drawing window.
4. At the Specify radius of sphere or [Diameter]: prompt, enter **3.125** and press Enter or enter **D** to specify the diameter of the sphere.
5. At the Enter number of longitudinal segments for surface of sphere <16>: prompt, enter **24** and press Enter.
6. At the Enter number of latitudinal segments for surface of sphere <16>: prompt, enter **24** and press Enter.
7. Set the Conceptual visual style (or another visual style) current and change the viewing angle so you're looking at the sphere from an isometric view.
8. Save the drawing with the name `Completed12-5.dwg` to the Chapter 12 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Dish and dome

*Dishes* and *domes* are essentially the bottom and top halves of a 3D mesh sphere (see Figure 12-12). You define a dish or dome in the same way that you do a 3D mesh sphere, except that you specify half as many latitudinal segments because you create only half a sphere. A dish is created below the working plane or the specified point, whereas a dome is created above the working plane or specified point. You might use a dish for a foot pad on an object that needs to be elevated or a dome as a cap on a hexagon nut or a screw head.

**Figure 12-12:**  
A dish and a dome, but no fork or spoon in sight.



You start the command to create a 3D mesh primitive of a dish or dome by entering **3d** at the command prompt and choosing the Dish or Dome options. Or enter at the command prompt **ai\_dish** for a dish or **ai\_dome** for a dome.

### Exercise 12-6: Creating a 3D Mesh Dome

For this exercise, open the drawing file `Exercise12-6.dwg` from the Chapter 12 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D mesh dome.

1. At the command prompt, enter **3d** and press Enter.

2. At the Enter an option [Box/Cone/DIsh/Dome/Mesh/Pyramid/Sphere/Torus/Wedge]: prompt, enter **do** to draw a dome.
3. At the Specify center point of dish: prompt, pick a point near the middle of the drawing window.
4. At the Specify radius of dish or [Diameter]: prompt, enter **2.75** and press Enter or enter **D** to specify the diameter of the sphere.
5. At the Enter number of longitudinal segments for surface of dish <16>: prompt, enter **12** and press Enter.
6. At the Enter number of latitudinal segments for surface of dish <8>: prompt, enter **6** and press Enter.
7. Set the Conceptual visual style (or another visual style) current and change the viewing angle so you are looking at the sphere from an isometric view.
8. Save the drawing with the name `Completed12-6.dwg` to the Chapter 12 folder in the My Practice Drawings folder under My Documents and close the drawing.

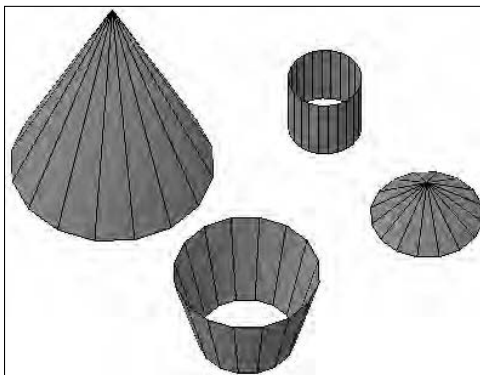


5. Exercise 12-6 shows how to create a dome. What steps do you take to create a dish?

1. \_\_\_\_\_
2. \_\_\_\_\_
3. \_\_\_\_\_
4. \_\_\_\_\_

## Cone

*Cones* are defined with a curved base and can have a top that forms a flat top or peak (see Figure 12-13). When you define a cone, you can create as many edges as you want at its base to create an octagon or even a hexagon shape, which allows you to create many different types of objects quickly. For example, you can define a cone with a top radius value that matches its base radius to create a three-dimensional bolt head. If you create a cone with no peak, the top is left open, as well as the bottom. If you want to close the object, you can draw a series of 3D faces or a mesh object. You also can't define a cone in the negative direction, so you would need to use the 3DROTATE command to flip it upside down if you need to.



**Figure 12-13:**  
Objects  
created  
from 3D  
mesh  
cones.



To start creating a 3D mesh primitive of a cone, enter **3d** at the command prompt and choose the Cone option or enter **ai\_cone** at the command prompt.



### Exercise 12-7: Creating a 3D Mesh Cone

For this exercise, open the drawing file `Exercise12-7.dwg` from the Chapter 12 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D mesh cone.

1. At the command prompt, enter **3d** and press Enter.
2. At the Enter an option [Box/Cone/DIsh/Dome/Mesh/Pyramid/Sphere/Torus/Wedge]: prompt, enter **c** and press Enter to draw a cone.
3. At the Specify center point for base of cone: prompt, pick a point near the middle of the drawing window.
4. At the Specify radius for base of cone or [Diameter]: prompt, enter **4.1675** and press Enter or enter **D** to specify the diameter of the cone.
5. At the Specify radius for top of cone of [Diameter] <0>: prompt, press Enter to create a point at the top of the cone.

If you enter a radius greater than 0 (zero), you create a cone with a flat top.

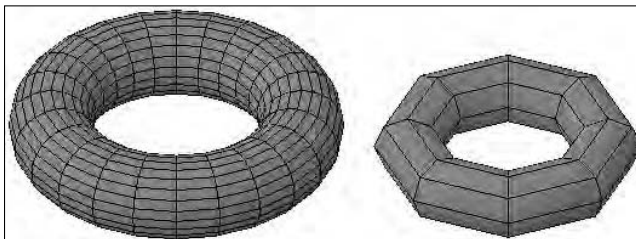
If you create a cone with the same radius for its top and base, you create a cylinder. This is the reason why there's no cylinder option for the 3D command. However, the object that you create is open on both the top and bottom ends, and you would have to use 3D faces or another 3D mesh object to close those ends.



6. At the Specify height of cone: prompt, enter **9.675** and press Enter.
7. At the Enter number of segments for surface of cone <16>: prompt, press Enter to accept the default number of segments.
8. Set the Conceptual visual style (or another visual style) current and change the viewing angle so you're looking at the sphere from an isometric view.
9. Save the drawing with the name `Completed12-7.dwg` to the Chapter 12 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Torus

A *torus* is a circular, tube-shaped object, like a doughnut, and is one of the least-used 3D mesh primitives that AutoCAD offers (see Figure 12-14).



**Figure 12-14:**  
Totally tubular toruses.

To start creating a torus, enter **3d** at the command prompt and choose the Torus option, or enter **ai\_torus** at the command prompt. When you create a torus, half of the object is created above and half below the current plane or point specified, just like a sphere is. You define the center of the torus, its outermost radius, the radius of the tube, and the number of segments around the circumferences of the tube and torus.

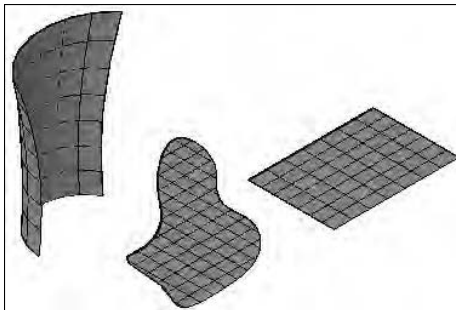
### Exercise 12-8: Creating a 3D Mesh Torus

For this exercise, open the drawing file `Exercise12-8.dwg` from the Chapter 12 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D mesh torus.

1. At the command prompt, enter **3d** and press Enter.
2. At the Enter an option [Box/Cone/DIsh/Dome/Mesh/Pyramid/Sphere/Torus/Wedge]: prompt, enter **T** and press Enter to draw a torus.
3. At the Specify center point of torus: prompt, pick a point near the middle of the drawing window.
4. At the Specify radius of torus or [Diameter]: prompt, enter **5** and press Enter or enter **D** to specify the diameter of the torus.
5. At the Specify radius of tube or [Diameter]: prompt, enter **1.25** and press Enter or enter **D** to specify the diameter of the tube.
6. At the Enter number of segments around tube circumference <16>: prompt, enter **24** and press Enter.
7. At the Enter number of segments around torus circumference <16>: prompt, enter **24** and press Enter.
8. Set the Conceptual visual style (or another visual style) current and change the viewing angle so you're looking at the sphere from an isometric view.
9. Save the drawing with the name `Completed12-8.dwg` to the Chapter 12 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Planar Surfaces and Surface Objects

Before you take the world by storm with your new-found surface modeling skills, you need to know about the *surface object*. Surface objects are relatively new in AutoCAD; they were introduced with AutoCAD 2007. Surface objects are commonly created through techniques such as lofting, extruding, and sweeping open 2D profiles, but you can also create a planar surface. Surface objects offer some of the functionality that 3D solids do for editing them, but the editing that you can perform is limited. Surface objects come into play when you need a plane to slice through 3D solids to make contoured faces. Figure 12-15 shows a surface object created with the LOFT command and a couple of planar surface objects.



**Figure 12-15:**  
Examples of  
surface  
objects.

## Planar surface

The planar surface object is a flat object that creates floors or adds a surface below a model to receive shadows that are being cast from light sources. You can create planar surfaces in two different ways:

- ✓ Pick two points in a drawing. Figure 12-15 shows a planar surface on the right, which was defined by picking two points.
- ✓ Select a closed 2D object such as a circle, closed polyline, or region.

When you select an object to create a planar surface from, the DELOBJ system variable controls what happens to the original object that you select. If you want to keep the original closed 2D object, you have to set the system variable DELOBJ to 0 (zero) before selecting the object to create a planar surface from. After you create a planar surface, it displays lines, which are its isolines, across the surface. You can control the number of isolines that are displayed by the current values of the system variables SURFU and SURFV before creating the planar surface, as I mention earlier in this chapter. You can also change the number of isolines displayed by using the Properties palette after selecting the planar surface object.

You can create a planar surface with any of these methods:

- ✓ Choose Draw⇒Modeling⇒Planar Surface.
- ✓ Click Planar Surface on the Modeling toolbar.
- ✓ Click Planar Surface on the 3D Make panel of the Dashboard.
- ✓ Enter **planesurf** at the command prompt.



Turn off orthomode when creating a planar surface object. For some reason, the command that creates the object doesn't like orthomode.

---

### Exercise 12-9: Creating a Planar Surface

For this exercise, open the drawing file `Exercise12-9.dwg` from the Chapter 12 folder in the My Practice Drawings folder under My Documents. Then practice creating a planar surface.

1. Make sure ORTHO is turned off before proceeding.
2. Choose Draw⇒Modeling⇒Planar Surface.
3. At the `Specify first corner or [Object] <Object>:` prompt, enter **0,6** and press Enter.
4. At the `Specify other corner:` prompt, enter **15,0** and press Enter.
5. Save the drawing with the name `Completed12-9.dwg` to the Chapter 12 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Complex surface objects

You can create surface objects (other than those created with the PLANESURF command) using some of the same commands that I introduce in Chapter 8 for creating complex 3D solids. Creating complex surface objects allows you to create much more elaborate surface models and can be used as slicing planes with the SLICE command on 3D solids. I cover creating surface objects with commands like LOFT and REVOLVE along with a few others in Chapter 13.

## *Answers to Practice Opportunities*

- 1. What steps do you think you take to create a 3D face?**
  1. Choose Draw→Modeling→Meshes→3D Face.
  2. Specify the points of the face at the prompts.
  3. Run the 3DFACE command again to create a second 3D face.  
The shared edge between the two faces should be invisible.
- 2. What option is available when creating a 3D mesh primitive of a box or wedge that is not available when creating a 3D solid primitive of a box or wedge?**

A. The only difference when creating a 3D mesh box or wedge or a 3D solid box or wedge is that when you create a 3D mesh box or wedge you can rotate the new object after specifying the dimensions of the primitive.
- 3. What steps do you take to create a 3D mesh primitive of a wedge?**
  1. Create a new drawing using the acad3d.dwt drawing template.
  2. Use the 3D command with the Wedge option.
  3. Specify the wedge's length, width and height.
  4. Specify the rotation angle of the wedge.
- 4. What steps do you take to create a three-sided pyramid (tetrahedron) with a top and not an apex?**
  1. Create a new drawing using the acad3d.dwt drawing template.
  2. Use the 3D command with the Pyramid option.
  3. Specify the three corners of the pyramid's base.
  4. Use the Tetrahedron option to create a pyramid with three-sides.
  5. Use the Top option and specify the three points that form the top of the pyramid.
- 5. What steps do you take to create a dish?**
  1. Create a new drawing using the acad3d.dwt drawing template.
  2. Use the 3D command with the Dish option.
  3. Specify the center point and radius for the dish.
  4. Specify the number of longitudinal and latitudinal segments for the dish.

## Chapter 13

# Complex Surface Modeling

### *In This Chapter*

- ▶ Creating complex 3D meshes
- ▶ Creating complex surfaces

Surface modeling isn't limited to just creating 3D faces, 3D mesh primitives, and planar surfaces. Prior to AutoCAD 2007, you could create complex 3D meshes based on picking a number of vertices to form a mesh, revolve 2D profiles around an axis, create a ruled mesh between two 2D objects, and a few other methods, which I talk about in this chapter. Starting with AutoCAD 2007, you could create not only planar surface objects, but complex surface objects through several different techniques. Some of the techniques that you can use to create complex surface objects include revolving and extruding open 2D profiles. The exercises in this chapter show you how to create both complex 3D meshes and surface objects.

## *Creating Complex 3D Meshes*

Complex 3D meshes allow you to complete a surface model with greater detail than just using 3D faces and mesh primitives. You can create complex 3D meshes through a number of different methods using existing 2D geometry that you might already have created in a drawing. You can create revolved, ruled, and tabulated meshes with existing 2D geometry. Another concept that you can use to create complex objects for a surface model is to create what is known as an *irregular mesh* (or nonplanar mesh).

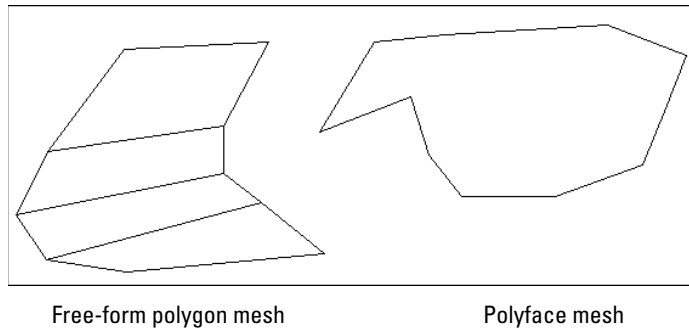
Don't be afraid to get your hands dirty: The following exercises get a bit meshy.

### *Irregular-shaped 3D meshes*

*Irregular meshes* blend areas between 3D mesh primitives that don't quite fit snugly together, or they can close an open mesh. AutoCAD allows you to create two different types of irregular meshes:

- ✓ **Free-form polygon:** This mesh has a number of segments running in the M and N directions. After you define the number of segments, you specify where each of the vertices for the mesh is to be placed in the drawing.
- ✓ **Polyface:** This mesh is like the free-form polygon mesh type in that it's defined vertex by vertex, but you don't need to define the number of segments that the mesh should be defined with. However, when you create a polyface mesh, you first define a series of vertices and then determine how the vertices are connected to create the mesh. Figure 13-1 shows a free-form polygon mesh on the left and a polyface mesh on the right. All of the 3D mesh primitives that I cover in Chapter 12 are polyface meshes.

**Figure 13-1:**  
A free-form  
polygon  
mesh and a  
polyface  
mesh.



Both the free-form polygon mesh and polyface mesh are usually used when customizing AutoCAD rather than for everyday modeling. Because the two irregular-shaped 3D meshes are primarily targeted for customizing, they have a learning curve to them. Many users don't use them and just create a series of 3D faces between points in a drawing to create irregular shapes if they need to.

You can create a free-form polygon mesh by entering **3dmesh** at the command prompt. You can create an irregular polyface mesh by entering **pface** at the command prompt.

Depending on the desired outcome, it can often be easier to create a planar 3D mesh object with the Mesh option of the 3D command and then manipulate the vertices of the mesh as needed to create the irregular 3D mesh that you need.



1. In the following exercise, you create a 3D mesh. What steps do you think you take to create a 3D mesh located on the top of a 3D model?

1. \_\_\_\_\_
2. \_\_\_\_\_
3. \_\_\_\_\_
4. \_\_\_\_\_

### Exercise 13-1: Capping a 3D Mesh with an Open Top Using a Free-Form Polygon Mesh



For this exercise, open the drawing file `Exercise13-1.dwg` from the Chapter 13 folder in the My Practice Drawings folder under My Documents. Then practice creating a free-form polygon mesh that you can use to cap the top of a 3D mesh.

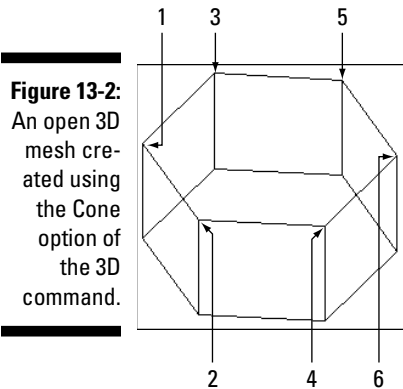
1. Turn on running object snap and set the Endpoint object snap current.



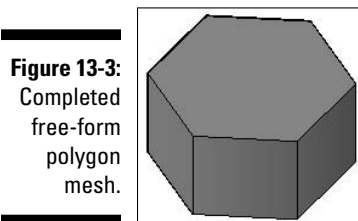
Right-click the OSNAP button on the status bar and choose Settings. In the Drafting Settings dialog box, Object Snap tab, click Object Snap On if it is not checked. Click the Clear All button and then click Endpoint. Click OK to save the changes and exit the Drafting Settings dialog box.

2. At the command prompt, enter **3dmesh** and press Enter.
3. At the Enter size of mesh in M direction: prompt, enter **3** and press Enter.
4. At the Enter size of mesh in N direction: prompt, enter **2** and press Enter.

- At the Specify location for vertex (0, 0): prompt, pick the end point indicated by 1 in Figure 13-2.



- Continue picking the endpoints at the points indicated by 2 through 6.
- Set the Conceptual visual style (or another visual style) current so you can see that the new 3D mesh object hides the inside of the 3D mesh. Figure 13-3 shows what the completed 3D mesh looks like.
- Save the drawing with the name `Completed13-1.dwg` to the Chapter 13 folder in the My Practice Drawings folder under My Documents and close the drawing.

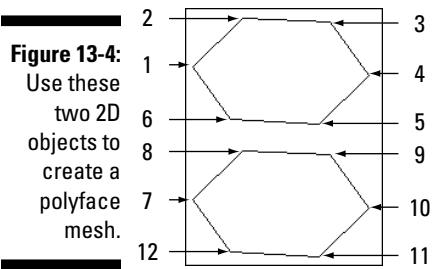


### Exercise 13-2: Creating a 3D Mesh from a Polyface Mesh

For this exercise, open the drawing file `Exercise13-2.dwg` from the Chapter 13 folder in the My Practice Drawings folder under My Documents. Then practice creating part of a hexagon shaped object from a polyface mesh.

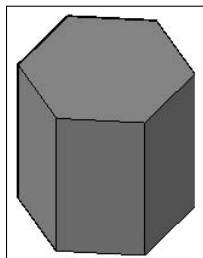
- Turn on running object snap and set the Endpoint object snap current.
- At the command prompt, enter **pface** and press Enter.
- At the Specify location for vertex 1: prompt, pick the end point indicated by 1 in Figure 13-4.
- Continue picking the endpoints at the points indicated by 2 through 12.
- At the Specify location for vertex 13 or <define faces>: prompt, press Enter.

You can now define each face that you want to create. To do so, you enter the vertices that you want to create a face between at the Enter a vertex number or [Color/Layer]: prompts. For example, to create a face that would be to the top of a hexagon shape, specify the numbers 1 through 6.



**Figure 13-4:**  
Use these two 2D objects to create a polyface mesh.

6. At the Face 1, vertex 1: Enter a vertex number or [Color/Layer]: prompt, enter **1** and press Enter.
7. At the Face 1, vertex 2: Enter a vertex number or [Color/Layer] <next face>: prompt, enter **2** and press Enter.
8. At the Face 1, vertex 3: Enter a vertex number or [Color/Layer] <next face>: prompt, enter **3** and press Enter.
9. At the Face 1, vertex 4: Enter a vertex number or [Color/Layer] <next face>: prompt, enter **4** and press Enter.
10. At the Face 1, vertex 5: Enter a vertex number or [Color/Layer] <next face>: prompt, enter **5** and press Enter.
11. At the Face 1, vertex 6: Enter a vertex number or [Color/Layer] <next face>: prompt, enter **6** and press Enter.
12. At the Face 1, vertex 6: Enter a vertex number or [Color/Layer] <next face>: prompt, press Enter to start creating the next face.  
 The next face you create is the one closest to you on the side of the hexagon-shaped object.
13. At the Face 2, vertex 1: Enter a vertex number or [Color/Layer]: prompt, enter **6** and press Enter.
14. At the Face 2, vertex 2: Enter a vertex number or [Color/Layer] <next face>: prompt, enter **5** and press Enter.
15. At the Face 2, vertex 3: Enter a vertex number or [Color/Layer] <next face>: prompt, enter **11** and press Enter.
16. At the Face 2, vertex 4: Enter a vertex number or [Color/Layer] <next face>: prompt, enter **12** and press Enter.
17. At the Face 2, vertex 5: Enter a vertex number or [Color/Layer] <next face>: prompt, press Enter to go to the next object.
18. Complete the remainder of the sides and bottom for the object. When you're done, your object looks similar to the one in Figure 13-5.



**Figure 13-5:**  
A completed polyface mesh.





The bad thing about using the PFACE command is that if you make a mistake you have to start all over again. Or instead of starting over, you can explode the mesh and remove the incorrectly drawn faces and refinish the mesh with the PFACE command or draw individual 3D faces using the 3DFACE command.

19. Save the drawing with the name `Completed13-2.dwg` to the Chapter 13 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Revolved meshes

Creating a revolved mesh is very similar to creating a revolved 3D solid or surface. (I cover creating revolved 3D solids in Chapter 8, and I discuss creating revolved surfaces later in this chapter.)

To create a revolved mesh, choose `Draw` → `Modeling` → `Meshes` → `Revolved Mesh` or enter **refsurf** at the command prompt. You need to have an open or closed 2D profile and know the axis that you want the profile to revolve around. You then specify the start angle and the number of degrees that the profile should be revolved. You can use 2D objects such as lines, circles, and arcs to create a revolved mesh, but using a 2D polyline is easier because you can only select a single object to revolve. Revolved meshes are great for representing objects such as pulleys, hubs, and even bolt heads.

When you create a revolved mesh, the system variables SURFTAB1 and SURFTAB2 control its smoothness:

- ✓ SURFTAB1 controls the number of times the profile is rotated around the specified axis.
- ✓ SURFTAB2 determines the number of segments that the profile should be equally divided into. The segments are divided perpendicular to the rotation axis.

To change the value of either SURFTAB1 or SURFTAB2, enter the variable's name at the command prompt and then enter the new value. Figure 13-6 shows a 2D polyline before and after being revolved to create a 3D mesh.



2. In the following exercise, you create a 3D mesh by revolving a closed 2D object. Which of the following is a closed 2D object?
  - A. Circle
  - B. Arc
  - C. Ellipse

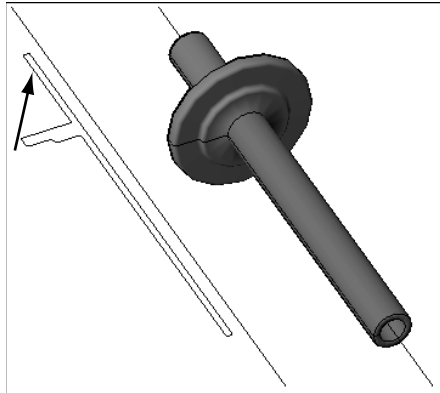


### Exercise 13-3: Revolving a Closed 2D Object to Create a 3D Mesh

For this exercise, open the drawing file `Exercise13-3.dwg` from the Chapter 13 folder in the My Practice Drawings folder under My Documents. Then practice creating a revolved 3D mesh.

1. At the command prompt, enter **surftab1** and press Enter.
2. At the Enter new value for SURFTAB1 <6>: prompt, enter **16** and press Enter.

3. Change the value of the system variable SURFTAB2 to 12.
4. Choose Draw→Modeling→Meshes→Revolved Mesh.
5. At the `Select object to revolve:` prompt, select the 2D polyline profile indicated by the arrow in Figure 13-6.
6. At the `Select object that defines the axis of revolution:` prompt, select the line to the right of the 2D polyline profile in Figure 13-6.



**Figure 13-6:**  
A revolved  
3D mesh  
from a 2D  
polyline.

7. At the `Specify start angle <0>:` prompt, press Enter to accept the default start angle of 0 (zero).
  8. At the `Specify included angle (+ccw, -cw) <360>:` prompt, press Enter to revolve the 2D profile a full 360 around the axis.
- The completed revolved mesh looks like the one on the right in Figure 13-6.
9. Save the drawing with the name `Completed13-3.dwg` to the Chapter 13 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Tabulating meshes

Tabulating meshes has nothing to do with counting ballots or elections, but rather creating a mesh, which is similar to creating an extruded 3D solid. To create a tabulated mesh, you start with an open or closed 2D profile such as a line, a circle, an arc, or a 2D polyline. I recommend using 2D polylines because you can select only a single profile to create a tabulated mesh from. Then choose Draw→Modeling→Meshes→Tabulated Mesh or enter `tabsurf` at the command prompt. After you've selected the 2D profile, you select an object to determine the vector in which the object should be drawn. Unlike the EXTRUDE command, you can't just specify a height to create the mesh. A tabulated mesh is great for creating shafts or rails.

When you create a tabulated mesh, the system variable SURFTAB1 controls its smoothness. It controls the number of segments that the 2D profile is equally divided into. To change the value of SURFTAB1, you enter the variable's name at the command prompt and then enter the new value when prompted. Figure 13-7 shows the results of creating a tabulated 3D mesh from the closed spline.

### Exercise 13-4: Creating a Tabulated 3D Mesh

For this exercise, open the drawing file `Exercise13-4.dwg` from the Chapter 13 folder in the My Practice Drawings folder under My Documents. Then practice creating a tabulated 3D mesh.

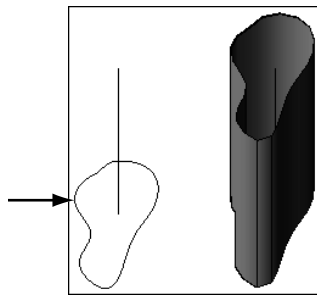


1. Change the value of the system variable `SURFTAB1` to 24.

You can change the value of a system variable by entering its name at the command prompt, and then entering the new value. To change the value of the `SURFTAB1` system variable, enter **surftab1** at the command prompt and press Enter. When prompted for the new value, enter the new value and press Enter.

2. Choose `Draw` → `Modeling` → `Meshes` → `Tabulated Mesh`.
3. At the `Select object for path curve:` prompt, select the 2D spline profile indicated by the arrow in Figure 13-7.

**Figure 13-7:**  
A tabulated  
3D mesh  
from a  
closed  
spline.



4. At the `Select object for direction vector:` prompt, select near the bottom of the line that is going in the Z direction closest to the spline.  
  
The end of the path that you select controls which direction the tabulated mesh is created in. The completed tabulated mesh looks like the one on the right in Figure 13-7.
5. Save the drawing as the name `Completed13-4.dwg` to the Chapter 13 folder in the My Practice Drawings folder under My Documents and close the drawing.

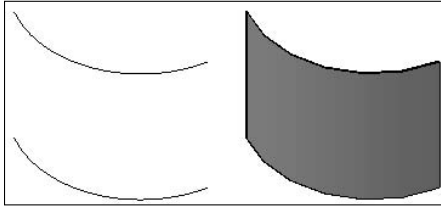
## One mesh to rule them all

Ruled meshes aren't subjects to a mesh named King Mesh or anything along those lines. To get a ruled mesh, you select two 2D profiles (open or closed), and AutoCAD calculates the segment points that fall on both objects to create a mesh. Usually you create ruled meshes from curved objects because they're the hardest to quickly create a good-looking mesh between. To create a ruled mesh, you can select two 2D profiles (open or closed) that are on the same or different planes.

Both objects that make up the ruled mesh must be either open or closed; you can't use one of each. In addition to using open and closed 2D objects, you can use a point object in place of one of the 2D objects to have the mesh converge to a single vertex. You can select only one point object when creating a ruled mesh, but it can be either the first or second object selected. You can create a ruled mesh for a hole that goes through a part or an inset. Ruled meshes are good to use instead of tabulated meshes when you have two different profiles to work with that might be of the same or different sizes.

You create a ruled mesh between two 2D objects by choosing Draw→Modeling→Meshes→Ruled Mesh or entering **rulesurf** at the command prompt. The system variable SURFTAB1 controls its smoothness. The system variable SURFTAB1 controls the number of segments that each of the two 2D profiles is equally divided into. To change the value of SURFTAB1, enter the variable's name at the command prompt and then enter the new value. Figure 13-8 shows the results of creating a ruled 3D mesh from the two 2D objects.

**Figure 13-8:**  
A ruled 3D  
mesh  
between  
two 2D  
objects.



### Exercise 13-5: Creating a Ruled 3D Mesh



For this exercise, open the drawing file `Exercise13-5.dwg` from the Chapter 13 folder in the My Practice Drawings folder under My Documents. Then practice creating a ruled 3D mesh.

1. Copy the arc in the drawing up 6 units in the Z direction.
2. Change the value of the system variable SURFTAB1 to 8.
3. Choose Draw→Modeling→Meshes→Ruled Mesh.
4. At the `Select first defining curve:` prompt, select near the end point of the arc of the bottom on the right side.
5. At the `Select second defining curve:` prompt, select near the end point of the arc of the top on the right side.

The completed ruled mesh looks like the one on the right in Figure 13-8.

6. Save the drawing with the name `Completed13-5.dwg` to the Chapter 13 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Edgy meshes

You can create meshes based on four adjoining edges that form a closed object. Creating meshes based on four edges allows you to make more organic-looking objects, such as fabrics or liquids that are in a container. You can also create meshes between four edges to help finish off part of a surface model that might need to be capped or to blend other faces together. The four objects that you select to create the mesh must start or end at the end point of the adjoining edge, and they must be a line, an arc, a spline, or a polyline. The first object that you select determines the M direction of the mesh, and the adjoining edges that touch the endpoints of the first object determine the N direction.

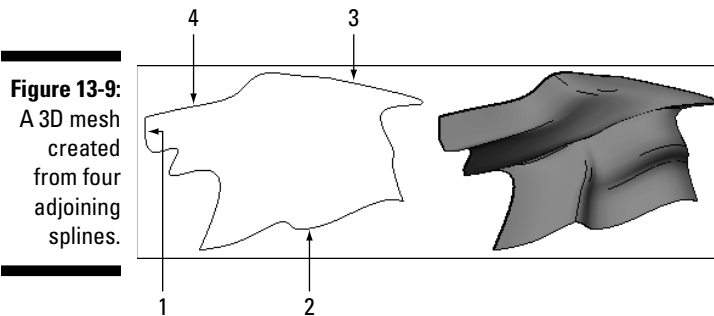
To start a mesh patch between four adjoining edges, choose Draw→Modeling→Meshes→Edge Mesh or enter **edgesurf** at the command prompt.

The number of segments that run in the M and N directions determines how smooth the transitioning mesh is between the selected four objects. The SURFTAB1 system variable determines the number of segments in the M direction, and the SURFTAB2 system variable determines the number of segments in the N direction. To change the value of either SURFTAB1 or SURFTAB2, enter the variable's name at the command prompt and then enter the new value when prompted. Figure 13-9 shows a 3D mesh created by selecting four adjoining edges that are defined splines. Notice that each spline ends and starts at the endpoint of the adjoining spline.

### Exercise 13-6: Creating a 3D Mesh Based on Four Adjoining Edges

For this exercise, open the drawing file `Exercise13-6.dwg` from the Chapter 13 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D mesh based on four adjoining edges.

1. Change the value of both the system variables SURFTAB1 and SURFTAB2 to 24.
2. Choose Draw→Modeling→Meshes→Edge Mesh.
3. At the `Select object 1 for surface edge:` prompt, select the edge indicated by 1 in Figure 13-9.



**Figure 13-9:**  
A 3D mesh  
created  
from four  
adjoining  
splines.

4. At the `Select object 2 for surface edge:` prompt, select the edge indicated by 2.
5. At the `Select object 3 for surface edge:` prompt, select the edge indicated by 3.
6. At the `Select object 4 for surface edge:` prompt, select the edge indicated by 4.

The completed mesh looks like the one on the right in Figure 13-9.

7. Save the drawing with the name `Completed13-6.dwg` to the Chapter 13 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Creating Complex Surfaces

Creating complex surfaces isn't too much different from creating complex 3D solids. The main difference is that you can't use the commands UNION, SUBTRACT, INTERSECT, and INTERFERE on surfaces to combine and remove sections from other objects in your model, so you have to do some additional planning upfront. However, techniques like lofting and revolving are available for creating complex surfaces.

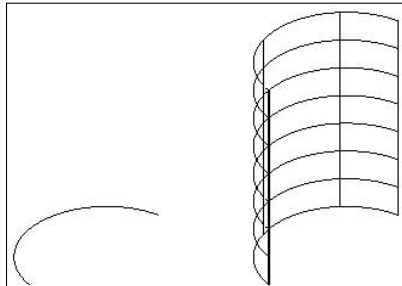
Because surface objects aren't created from straight segments like complex 3D meshes, the objects that you create as surface objects appear smooth rather than blocky. In these sections, I look at the techniques that you can use to create complex surfaces.



When creating complex surface objects, the DELOBJ system variable controls — in most cases — whether the selected 2D profiles that you pick are kept in the drawing or are deleted. Set DELOBJ to 0 (zero) to keep the original 2D profiles that are selected after the command ends; otherwise, by default they're deleted.

## Extruded surfaces

Extruding surfaces are very similar to creating extruded 3D solids. To create an extruded surface, you first select an open 2D profile, not a closed 2D profile. Unlike creating a tabulated 3D mesh, you can specify a height, direction, or path when creating an extruded surface from a 2D object. While extruding the 2D object, you can apply a taper if you need to. To create an extruded surface object, you use the EXTRUDE command. (To find out more about how to use the EXTRUDE command, see Chapter 6.) Figure 13-10 shows an open 2D profile before and after using the EXTRUDE command.

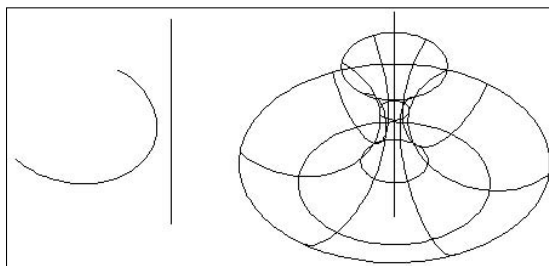


**Figure 13-10:**  
An example  
of an  
extruded  
surface.

## Revolved surfaces

You create revolved surfaces the same way that you create a revolved 3D solid, except that you use an open profile instead of a closed profile. A revolved surface is much better looking than a 3D mesh that you create with the REVSURF command because a revolved surface appears curved instead of blocky, as the mesh does.

To create a revolved surface, you use the REVOLVE command. After the command is started, select the open 2D profile and then the axis that it should be revolved around. You can specify the axis by using the current X, Y, or Z axis; by using an object; or by using an axis defined by two points. (To find out more about how to use the REVOLVE command, see Chapter 8.) Figure 13-11 shows an open 2D profile revolved around a line that's used as the axis for the revolution of the profile.

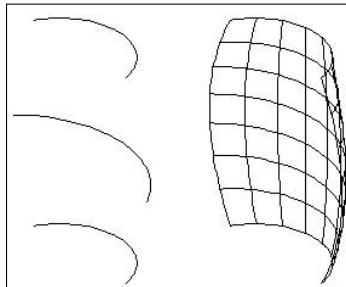


**Figure 13-11:**  
An example  
of a  
revolved  
surface.

## Lofted surfaces

You can create surface objects with the lofting technique. To create a lofted surface object, you need to create two or more open profiles. Then you need to select each profile in the order that you want the lofting to happen. After that, you can determine whether the lofted object is created using only the selected cross-sections, by selecting a path for the lofted surface to follow, or by using guides to define the surface object.

The LOFT command creates a lofted surface, and the process is just like creating a lofted 3D solid. There is no command to create a 3D mesh that's equivalent to the LOFT command, other than creating multiple cross sections and using the RULESURF command. (To find out more about how to use the LOFT command, see Chapter 8.) Figure 13-12 shows a lofted surface created from several open 2D profiles.

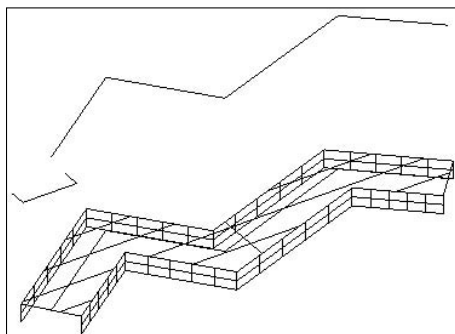


**Figure 13-12:**  
An example  
of a lofted  
surface.

## Swept surfaces

Swept surfaces are created in a same way that a swept 3D solid is created. To create a swept surface, you must have an open 2D profile and a path that are both created from 2D objects. Select the profile and then the path to cast the profile along to create a swept surface. Additionally, you can adjust the base point and alignment of the 2D profile with the path, and you can specify a scale and twist value for the 2D profile as it is being cast to create the surface object.

There is no command to create a 3D mesh that's equal to the SWEEP command. With some patience and some ingenuity, you can use the DIVIDE command to align 2D profiles with a path and then use the RULESURF command to kind of simulate a swept 3D mesh. To find out more about how to use the SWEEP command, see Chapter 8. Figure 13-13 shows a swept surface created from an open 2D profile cast along a path.



**Figure 13-13:**  
Example  
of a swept  
surface.

## *Converting objects to surfaces*

Starting with AutoCAD 2007, you can convert 2D objects to surface objects by using the CONVTSURFACE command (Modify↔3D Operations↔Convert to Surface). To create surfaces from 2D objects, select one of the 2D objects compatible with the command. The objects that you can select must either be closed or have a thickness assigned to them. The CONVTSURFACE command allows you to convert the following objects to surface objects:

- ✓ 2D solids
- ✓ Regions
- ✓ Open 2D polyline with thickness and no width
- ✓ Lines and arcs with thickness
- ✓ Planar 3D faces

You can also convert 2D objects into 3D solids by using the CONVTSOLID command. To discover more about the CONVTSOLID command, see Chapter 8.

## *Answers to Practice Opportunities*

### **1. What steps do you think you take to create a 3D mesh located on the top of a 3D model?**

1. Create a new drawing using the acad3d.dwt drawing template.
2. Start the 3DMESH command.
3. Create a 3D mesh that measures a size of 3 in the M direction and 2 in the N direction.
4. Specify the six points in the drawing to create the 3D mesh.

### **2. Which of the following is a closed 2D object?**

A, C. A closed 2D object is an object that doesn't have any openings (or gaps) and is one continuous object. Examples of closed 2D objects are circles, ellipses, closed 2D polylines, and closed 2D splines. While you can draw lines and arcs to create a closed boundary, they're not a single object. You can convert lines and arcs that form a closed boundary to a closed 2D polyline if the lines and arcs touch end to end.



## Chapter 14

# Modifying Surfaces

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### *In This Chapter*

- ▶ Editing surface models in general
  - ▶ Editing 3D faces
  - ▶ Editing 3D meshes
  - ▶ Editing surface objects
- 

**A**lthough surface models are limited in how they can be edited, you can use several different methods to modify 3D faces, 3D meshes, and surface objects. You edit surface models by using commands such as 3DMOVE, COPY, 3DROTATE, and most of the other modify commands, but few commands are specific to editing surface models (although there are plenty for 3D solid models). The commands FILLET and CHAMFER don't have additional options for editing surfaces like they do with 3D solids, but the PEDIT command does hold a secret for editing 3D meshes. The exercises in this chapter show you how to modify 3D faces, 3D meshes, and surface objects.

## *Editing Surface Models*

You edit surface models by using most of the standard editing techniques that you use to modify 2D objects and even 3D solids. One of the most powerful ways to modify objects in a drawing is through the use of grips; you can use grips to move, rotate, copy, and scale objects that create surface models. For example, you can stretch objects like 3D faces and meshes by using grips. When stretching a 3D face or mesh with grips, you're moving the vertices of the object or objects. By using grips with 3D faces and meshes, you can quickly add a feature like tapering between surfaces. Surface objects can't be deformed like 3D faces, but meshes can be deformed because they aren't defined by a series of vertices.

Go to the Properties palette (Tools⇨Palettes⇨Properties) to modify the properties and vertices of 3D faces, 3D meshes, and surface objects. You can modify general properties such as color, layer, or material, along with object-specific properties like the location of a vertex and an object's height. The properties that are displayed in the Properties palette depend on what type of object is selected and the number of objects that are selected.

## *Editing 3D Faces*

As you create a 3D face, you determine where each endpoint of the object is created and which sides remain visible. It's often easier to just worry about creating the 3D face first and then determine which edges should be visible. You can adjust the endpoints of a 3D face by using grips of the properties Vertex, Vertex X, Vertex Y, and Vertex Z in the Properties palette. The Vertex property allows you to designate which vertex of the 3D face you want to work with. The selected vertex is indicated in the drawing by an X.

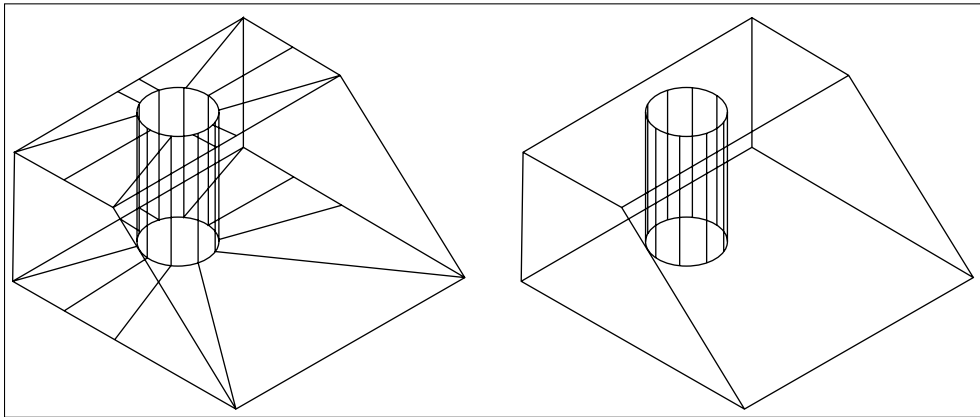
Using grips to modify the location of a vertex for a 3D face is easier than using the Properties palette. Along with editing the vertices of a 3D face and its general properties by using grips and the Properties palette, you can change which edges are visible.

When creating 3D faces, it takes time to figure out exactly which edges you want visible and which ones you want hidden. If you didn't take the time to figure this out when you created your 3D face, you can use the EDGE command to change the visibility of the edges.

The EDGE command allows you to interactively select the edges on a 3D face or shared edges between two 3D faces to mark both the overlapping edges invisible. The EDGE command can also redisplay an invisible edge of a 3D face. Choose Draw → Modeling → Meshes → Edge or enter **edge** at the command prompt and then select the edges you want to mark as invisible. The invisible edge isn't displayed in the drawing unless you set the SPLFRAME system variable to 1. Then you have to regenerate the drawing (View → Regen All).

Figure 14-1 shows a surface model created with 3D faces. The image on the left shows all edges for the 3D faces visible, whereas the one on the right shows select adjoining edges on the top and bottom of the model set invisible to give the surfaces a cleaner look.

**Figure 14-1:**  
A surface model created with 3D faces that are displayed with (left) and without (right) hidden edges.



The Properties palette allows you to change the visibility for each edge of a 3D face, but it can be a little hard to tell which edge you're about to make invisible because you just choose from the properties Edge 1, Edge 2, Edge 3, and Edge 4 under the Geometry category. Controlling the visibility of the edges for the 3D faces through the palette doesn't really make sense unless you want all the edges for the selected 3D faces to be visible or invisible.



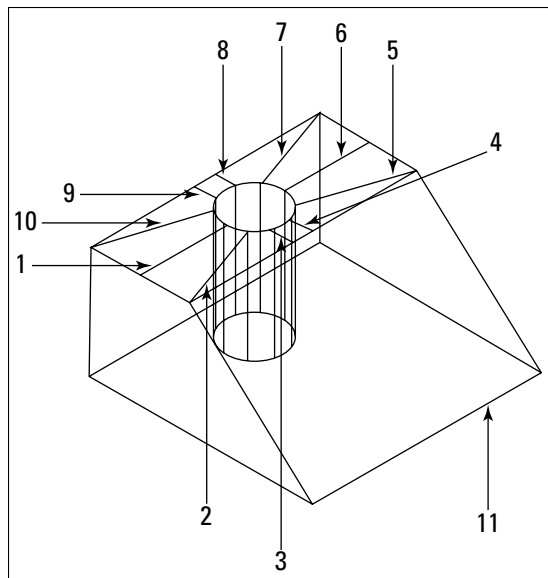
1. In Exercise 14-1, you change the visibility of the edges for 3D faces. What steps do you take to create a 3D face?
  1. \_\_\_\_\_
  2. \_\_\_\_\_
  3. \_\_\_\_\_



### Exercise 14-1: Hiding Edges of 3D Faces

For this exercise, open the drawing file `Exercise14-1.dwg` from the Chapter 14 folder in the My Practice Drawings folder under My Documents. Then practice making edges of a 3D face invisible and displaying hidden edges.

1. Choose `Draw` → `Modeling` → `Meshes` → `Edge`.
2. At the `Specify edge of 3dface to toggle visibility or [Display]:` prompt, select the line indicated by 1 in Figure 14-2.



**Figure 14-2:**  
Using the  
EDGE  
command  
to mark  
selected  
edges  
invisible.

Don't select an edge more than once; otherwise, you might inadvertently make it visible.

3. Continue selecting the edges marked 2 through 10 and press `Enter` when you've picked the last edge to end the command.
4. With no command active, select the two 3D faces that met at the edge indicated by 11 in Figure 14-2 using a crossing window. Pick a point below and to the right of the edge and then pick a point above and to the left of the edge to select both edges.
5. Display the Properties palette (choose `Tools` → `Palettes` → `Properties`). In the Properties palette, select `Hidden` from the Edge 4 property.

The model now appears incomplete because the edge that was indicated by 11 is invisible.

6. Restart the `EDGE` command. At the `Specify edge of 3dface to toggle visibility or [Display]:` prompt, enter `D` and press `Enter`.

You see the display mode of the `EDGE` command, which allows you to redisplay edges of 3D faces.

7. At the `Enter selection method for display of hidden edges` [`Select/All`] <`All`>: prompt, enter `A` and press `Enter`.

All the edges in the drawing now appear as dashed lines, which allows you to select them. You can use the `Select` option to select 3D faces in the drawing that you want to see the invisible edges for.

8. At the Specify edge of 3dface to toggle visibility or [Display]: prompt, pick the line indicated by 11 in Figure 14-2 and press Enter to end the command.

The edge is now visible once again.

9. At the command prompt, enter **splframe** and press Enter.
  10. At the Enter new value for SPLFRAME <0>: prompt, enter **1** and press Enter.
- The invisible edges of the 3D faces are now allowed to be displayed in the drawing, but won't be displayed until the current viewport is regenerated.

11. Choose View↔Regen or View↔Regen All.

The invisible edges of the 3D faces are now displayed in the current viewport. The difference between the Regen and Regen All option is that Regen performs a regeneration only on the current viewport, whereas Regen All regenerates the current viewport and all other viewports in the drawing. The drawing looks like the image on the left in Figure 14-1.

12. Change SPLFRAME back to a value of 0 (zero) and then regenerate the drawing.
- The model looks like the image on the right in Figure 14-1.
13. Save the drawing with the name `Completed14-1.dwg` to the Chapter 14 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Editing 3D Meshes

Modifying 3D meshes is slightly different from 3D faces, and it varies based on the type of 3D mesh you're modifying. You can modify polygon meshes in a few additional ways that aren't supported for polyface meshes. You create polyface meshes by using the PFACE command, and you create polygon meshes by using either the 3DMESH command or one of the options of the 3D command. I cover creating polygon meshes by using the 3D command in Chapter 12 and the 3DMESH command in Chapter 13. You find out about creating polyface meshes in Chapter 13.

You can use the standard modifying commands when editing 3D meshes, with the exceptions of commands like BREAK, TRIM, EXTEND, FILLET, and CHAMFER. Commonly, you're moving (or stretching) the vertices of a mesh using either grip editing or the Properties palette. Like grip editing with 3D faces, you can quickly modify all edges where vertices intersect using grips. You can also adjust the vertices of a 3D mesh by using the properties Vertex, Vertex X, Vertex Y, and Vertex Z in the Properties palette. The Vertex property allows you to designate which vertex on the mesh you want to work with. The selected vertex is indicated in the drawing by an X.



2. The \_\_\_\_\_ and \_\_\_\_\_ directions of a 3D mesh object control the number of vertices.

The Properties palette allows you to see the number of vertices used to create a 3D mesh, but you can't edit these values. The only way to add new vertices to a 3D mesh is to start over and create a completely new 3D mesh based on new coordinate points. Additionally, you can smooth a polygon mesh so it appears more contoured and break down a mesh into multiple objects.

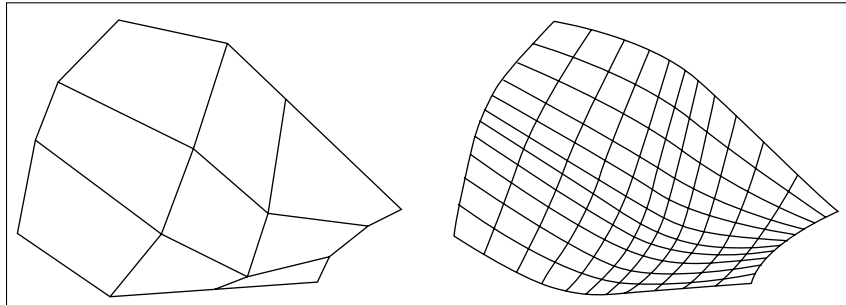
## Smoothing a mesh

Meshes are created by defining a number of vertices in both the M and N directions for polygon meshes or through a series of vertices, which define the surfaces of a polyface mesh. Polyface meshes are limited to the points you defined before determining how the mesh is created. But you can smooth polygon meshes — like their not-so-distant cousin, the polyline — to give them a more curved look. To smooth a polygon mesh, you can use either the PEDIT command or the Properties palette.

When you want to smooth a polygon mesh, you can choose three different surface-smoothing options: Quadratic, Cubic, or Bezier. Each option affects the 3D mesh in a slightly different way. When you apply one of the smoothing options, you can change the density that the mesh is created with. The PEDIT command allows you to smooth a polygon mesh but doesn't allow you to specify a smoothing option. The SURFTYPE system variable controls the type of smoothing that is used with the PEDIT command. If SURFTYPE is set to a value of 5, Quadratic smoothing is used. A value of 6 for SURFTYPE indicates Cubic smoothing, whereas a value of 8 indicates that Bezier smoothing should be used to smooth a polygon mesh.

If you change the smoothing option for a polygon mesh by using the Properties palette, you can specify a new density for the mesh to help make its edges appear curved from a distance instead of straight. To make the mesh denser and appear more curved, you adjust the M Density and N Density properties of the mesh. Figure 14-3 shows a mesh before and after the mesh's density is changed and Quadratic smoothing is applied.

**Figure 14-3:**  
The results  
of smooth-  
ing a poly-  
gon mesh  
using the  
Properties  
palette.



## A mesh no more

Meshes are complex objects like polylines, regions, and 3D solids. Like other complex objects, AutoCAD can break down certain objects into more basic 2D and 3D objects. For example, you can explode a polyline into lines and arcs or a 3D solid into regions. Breaking down a complex 3D object is no different from a complex 2D object: You use the EXPLODE command (Modify→Explode). When you select a 3D mesh at the *Select objects:* prompt for the EXPLODE command, AutoCAD takes a look at the mesh and calculates which edges are visible or invisible and creates individual 3D faces based on the vertices of the 3D mesh.

When you explode a polygon mesh, AutoCAD creates four-sided 3D faces, and all edges are visible. Exploding a polyface mesh results in the creation of both three- and four-sided 3D faces. The 3D faces that are created after exploding a polyface mesh retain their edge visibility settings, so if the edge was invisible before the 3D mesh was exploded, it becomes invisible after you explode it.



3. Exercise 14-2 shows how to modify a polygon mesh. What steps do you take to create a polygon mesh?

1. \_\_\_\_\_
2. \_\_\_\_\_
3. \_\_\_\_\_

---

### Exercise 14-2: Modifying a Polygon Mesh

For this exercise, open the drawing file `Exercise14-2.dwg` from the Chapter 14 folder in the My Practice Drawings folder under My Documents. Then practice applying the Quadratic smoothing option to a polygon face mesh and change the meshes density to make it smoother.

1. Select the polygon mesh and display the Properties palette.
2. On the Properties palette, select the Fit/Smooth property located under the Misc category near the bottom of the Properties palette.
3. From the drop-down list that appears, select Quadratic.

The polygon mesh doesn't appear as blocky along the edges, and the mesh is much denser.

4. Click in the text box to the right of the M Density property in the Properties palette. Enter a new value of **12** and press Enter.
  5. Change the N Density property to a value of 12.
  6. Press Esc to remove the grips from the 3D mesh so you can get a better look at it.
7. Select the polygon mesh once again and this time explode it by using the EXPLODE command.
  8. Try selecting the polygon mesh after it has been exploded. You only select one of the newly created 3D faces instead.

The drawing now looks like the 3D mesh on the right side of Figure 14-3.

9. Save the drawing with the name `Completed14-2.dwg` to the Chapter 14 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Editing Surface Objects

Modifying surface objects is much different from modifying 3D faces or meshes. You can't use grip editing to adjust the vertices of the mesh that is displayed on-screen because a surface object has no vertices. The lines that you see are the isolines of the object that help you identify whether the object is curved or flat. You can change the number of isolines that are displayed by using the Properties palette. Along with isolines affecting the display of a surface object, the VIEWRES command and FACETRES system variable also affect how a surface object appears in the drawing. (To find out more about VIEWRES and FACETRES, see Chapter 6.)

The Properties palette allows you to change properties that define dimensional characteristics of a surface object. For example, if you extrude an arc, you can change the

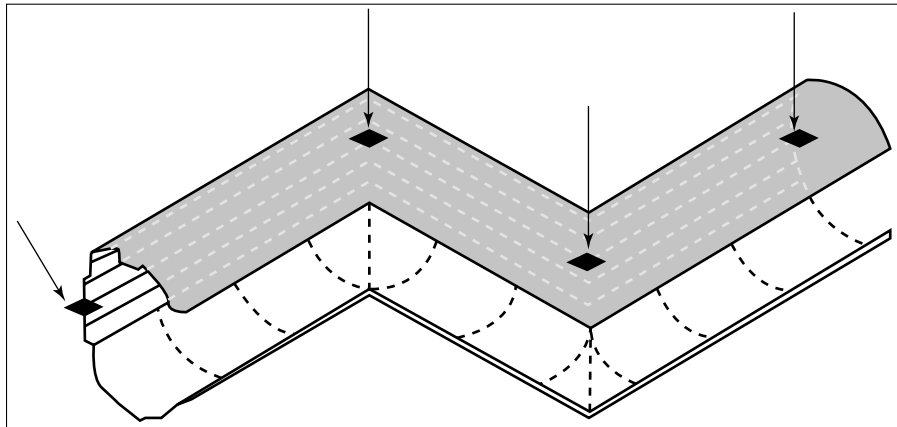
extruded surface's height and taper angle with the Properties palette. Each complex surface object has a different set of properties that appears in the Geometry category on the Properties palette.

## Reshaping a surface

Although it isn't possible to adjust the vertices of a surface object like you can with 3D faces or meshes, you can adjust dimensional characteristics of a surface object by using grips (such as its height if it's an extruded surface) or by using the original object that you used to create the surface object. So if an arc is extruded to create a channel, you can go back and change the arc's radius or its start and end points. Being able to edit the original 2D profile is very similar to being able to update the history of a 3D solid.

To change the shape of a surface object, select a surface object when no command is active to display its grips. Then you can edit the original object just as if it was a 2D object by first selecting the grip and then entering a new value or location for the selected grip. Figure 14-4 shows a surface object created by sweeping a 2D polyline along a path; grips for both the path and the original 2D polyline are displayed. The grips on the 2D polyline that was swept along the path are displayed on the end of the 3D object, whereas the grips for the object used as the path are displayed along the 3D object and are indicated by the arrows in Figure 14-4.

**Figure 14-4:**  
Editing a surface object by using grips from the original objects used to create the surface object.



## Surfaces to wireframe

Exploding a surface object with the EXPLODE command doesn't quite produce the results that you might think — probably a 3D mesh of some sort or 3D faces. Actually, you get a wireframe of the surface object.



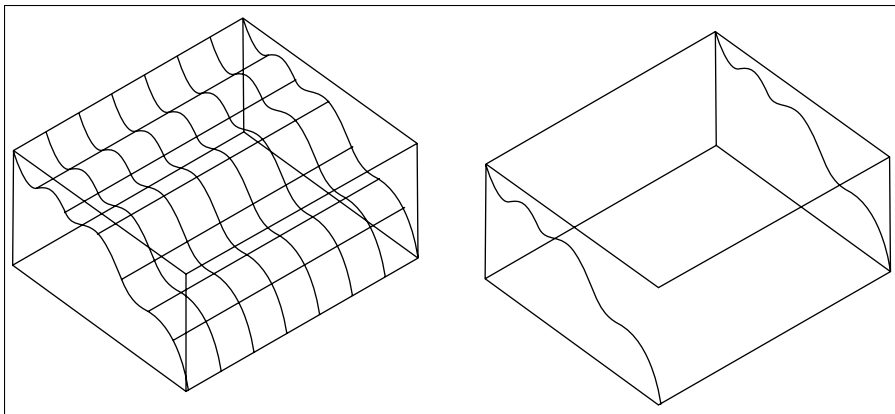
4. Which of the following best describes a wireframe model?
- A model that is created from 3D objects and displayed with the 2D Wireframe or 3D Wireframe visual style.
  - A model that is made up of 2D objects drawn at isometric angles.
  - A model that is made up of 2D objects that are drawn with a Z value greater than, equal to, or less than 0 (zero) to make the 2D objects appear as if they were 3D objects with the edges beyond not hidden.

When you have a wireframe model, you can create a mesh from the 2D objects created, use the 2D objects as construction lines to create 3D solids, or simply create a wireframe model. If you want to create a wireframe model from a surface object or 3D solid without using the EXPLODE command, you can use the XEDGES command to extract the visible edges of your model. The XEDGES command works on surface objects or 3D solids only; it doesn't work on 3D faces or meshes. To start the XEDGES command, enter **xedges** at the command prompt or choose Modify↔3D Operations↔Extract Edges.

## Slicing through 3D solids

Although the SLICE command doesn't allow you to modify surface objects, you can use surface objects as the slicing plane to shape 3D solids. To do so, make sure the surface object intersects the 3D solid. Start the SLICE command and select the 3D solids that you want to cut through. After you select the 3D solids to cut, use the Surface option to select a complex surface object that you created with the EXTRUDE, LOFT, SWEEP, or REVOLVE command. You can also use the Planar Surface option and select a planar surface object that you created with the PLANESURF command. After you cut the 3D solid, you need to select which part of the 3D solid to keep. Figure 14-5 shows a 3D solid before and after it's sliced with an extruded spline. (To find out more about the SLICE command, see Chapter 9.)

**Figure 14-5:**  
The results  
of using an  
extruded  
spline to  
shape a 3D  
solid.



### Exercise 14-3: Modifying a Surface Object



For this exercise, open the drawing file `Exercise14-3.dwg` from the Chapter 14 folder in the My Practice Drawings folder under My Documents. Then practice modifying a surface object that was created with the EXTRUDE command and use it as a slicing plane with the SLICE command.

1. Turn on object snap mode and enable the Endpoint running object snap.



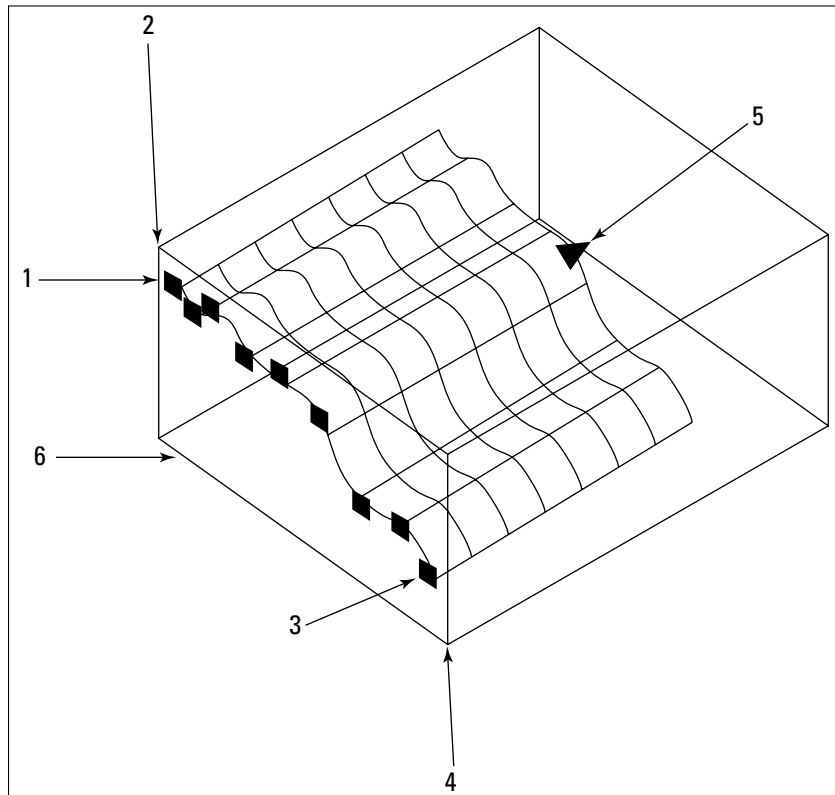
To turn on object snap mode, right-click the OSNAP button on the status bar and choose On from the shortcut menu. To enable the Endpoint running object snap, right-click the OSNAP button on the status bar and choose Settings from the shortcut menu. In the Drafting Settings dialog box, on the Object Snap tab, click the Clear All button and then click the Endpoint check box. Click OK to return to the drawing.

2. Select the surface object that's inside the box to display its grips.



3. Select the grip indicated by 1 in Figure 14-6 and select the end point of the 3D solid indicated by 2.

The grip at the point indicated by 1 is one of the end points of the spline used to create the extruded surface object.



**Figure 14-6:**  
Modifying a  
surface  
object with  
grips.

4. With the grips still displayed, select the grip indicated by 3 in Figure 14-6 and snap it to the end point of the 3D solid indicated by 4.
5. Turn on ortho mode and select the grip indicated by 5 in Figure 14-6.
6. With the grip selected, drag to the right and enter a value of 6. Press Enter to adjust the height of the surface object.
7. Start the SLICE command (Modify↔3D Operations↔Slice).
8. At the `Select objects:` prompt, select the 3D box and press Enter.
9. At the `Specify start point of slicing plane or [planar Object/Surface/Zaxis/View/XY/YZ/ZX/3points] <3points>:` prompt, enter **S** and press Enter to use the Surface option.
10. At the `Select a surface:` prompt, select the surface object.
11. At the `Select solid to keep or [keep Both sides] <Both>:` prompt, select the edge of the 3D solid indicated by 6 in Figure 14-6.

The bottom of the 3D solid is kept and so is the surface object. The SLICE command doesn't remove the surface object that's used as the cutting plane, so if you no longer need it, you can delete it.

- 12.** Set the Conceptual visual style current.

The 3D solid is now contoured and looks like the image on the right in Figure 14-5, with the exception that Figure 14-5 doesn't include a visual style.

- 13.** Choose **Modify**→**3D Operations**→**Extract Edges**.

- 14.** At the **Select objects:** prompt, enter **all** and press Enter twice.

AutoCAD creates a wireframe model from the surface and 3D solid in the drawing, but you most likely can't see them with the 3D solid and surface object displayed yet.

- 15.** Turn off the layer **Obj** to see the 2D objects that AutoCAD created.

The wireframe that's created from extracting the edges of the 3D solid and surface looks similar to the 3D model on the right side in Figure 14-5.

- 16.** Save the drawing with the name **Completed14-3.dwg** to the Chapter 14 folder in the My Practice Drawings folder under My Documents and close the drawing.

## *Answers to Practice Opportunities*

- 1. What steps do you take to create a 3D face?**

1. Start the 3DFACE command (choose **Draw**→**Modeling**→**Meshes**→**3D Face**).
2. When prompted to specify the corners of the 3D face, pick in the drawing window or enter coordinate values at the command prompt.
3. Specify a minimum of three points. Before specifying a point, remember to determine if the edge that you are creating for the 3D face should be hidden or not.

- 2. The M and N directions of a 3D mesh object control the number of vertices.**

- 3. What steps do you take to create a polygon mesh?**

1. Start the 3DMESH command (choose **Draw**→**Modeling**→**Meshes**→**3D Mesh**).
2. When prompted to, enter a size for the 3D mesh in the M and N directions.
3. Specify the location of each of the vertices for the 3D mesh.

- 4. Which of the following best describes a wireframe model?**

C. Wireframe models are created from 2D objects that are given a Z value that is greater than, equal to, or less than 0 (zero) and isn't capable of hiding objects beyond. Answer A is incorrect because a 3D object displayed with the 2D Wireframe or 3D Wireframe visual styles is still a 3D model that is capable of hiding objects. Answer B is wrong because drawing 2D objects at isometric angles might give the appearance of 3D objects, the drawing that is created is still flat.

## Chapter 15

# Additional Exercises: Modeling with Surfaces

### *In This Chapter*

- ▶ Getting more practice modeling with meshes
- ▶ Doing additional exercises focused on modeling with surfaces

**I**n the previous chapters of Part IV, you create and modify 3D faces, 3D meshes, and surface objects to create surface models in AutoCAD. You create surface models with varying techniques by using commands to create primitives such as a box or cone or to create complex meshes and surfaces by revolving 2D profiles. The exercises in this chapter continue the exercises in Chapters 7 and 11, and you create parts for the model by using 3D meshes and surface objects. Figure 15-1 shows the rear assembly of the custom motorcycle that you work with.



**Figure 15-1:**  
The rear  
assembly of  
a custom  
motorcycle.

Model courtesy of Mark Hechel of MasterGraphics, Inc.

## *Modeling with Meshes*

In this series of exercises, you create a 3D model of a bearing to be attached to the model of the rear assembly for the custom motorcycle. Like the model of the shock that you create in Chapter 11 by using 3D solids, the bearing is a purchased part. Because the part is being purchased, you would normally obtain the dimensional characteristics of the part from the vendor or actually measure a physical part yourself to create the 3D model.

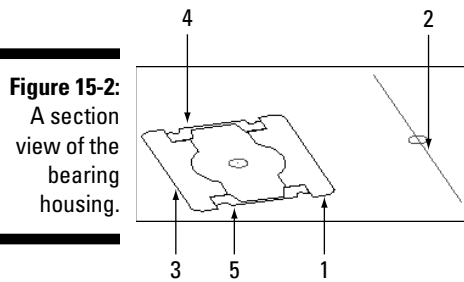


### Exercise 15-1: Creating the Outer Housing for a Bearing Using 3D Meshes

For this exercise, open the drawing file `Exercise15-1.dwg` from the Chapter 15 folder in the My Practice Drawings folder under My Documents. Then practice creating a complex 3D mesh by revolving a closed 2D profile around an axis to create the outer housing for the bearing.



1. Set the layer `Bearing_Housing_Outer` current.
2. Set the system variables `SURFTAB1` to 16 and `SURFTAB2` to 8.  
At the command prompt, enter the name of the system variable to change and then enter the new value when prompted for it.
3. Choose `Draw` → `Modeling` → `Meshes` → `Revolved Mesh`.
4. When prompted to specify the object to revolve, select the object indicated by 1 in Figure 15-2.

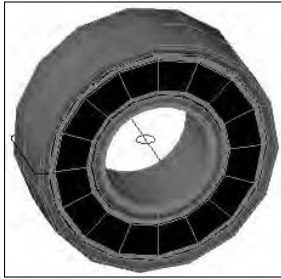


**Figure 15-2:**  
A section view of the bearing housing.

*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

5. At the `Select object that defines the axis of revolution:` prompt, select the line indicated by 2 in Figure 15-2.
6. Press `Enter` twice to accept the default starting angle and to revolve the 2D profile in a full 360 degrees.
7. Repeat the `REVSURF` command and select the 2D profile indicated by 3 in Figure 15-2. Use the same line that you previously selected as the axis to create the revolved mesh.  
The bearing is now starting to take shape.
8. Turn off the layer `Bearing_Housing_Outer` and set the layer `Bearing_Housing_Inner` current.
9. Repeat the `REVSURF` command two more times and select the 2D profiles indicated by 4 and 5 in Figure 15-2. Use the same line as the axis to create the revolved meshes.
10. Turn on the layer `Bearing_Housing_Outer` and set the `Realistic` visual style current.  
The model looks similar to the one shown in Figure 15-3.
11. Save the drawing with the name `Completed15-1.dwg` to the Chapter 15 folder in the My Practice Drawings folder under My Documents and close the drawing.

**Figure 15-3:**  
A section view of the bearing housing.



Model courtesy of Mark Hechel of MasterGraphics, Inc.

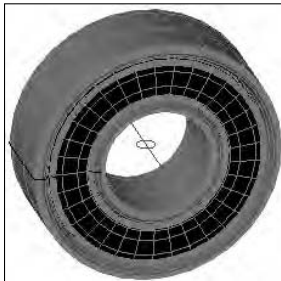
### Exercise 15-2: Smoothing the 3D Mesh for the Outer Housing of the Bearing

For this exercise, open the drawing file `Exercise15-2.dwg` from the Chapter 15 folder in the My Practice Drawings folder under My Documents. Then practice changing the smoothing option and mesh density of the revolved meshes that you created in the previous exercise to make them appear smoother.

1. Select the four polygon meshes that you created in Exercise 15-1.
2. Display the Properties palette and select Quadratic from the Fit/Smooth property.
3. Set the M Density to a value of 34 and N Density to a value of 42.

As you change the values, they might change in the Properties palette because of the original values used for the `SURFTAB1` and `SURFTAB2` system variables when creating the revolved meshes. The updated revolved meshes look like the ones in Figure 15-4. Refer to Figure 15-3 to see what the revolved meshes looked like before you smoothed them.

**Figure 15-4:**  
Revolved 3D meshes smoothed using Quadratic smoothing and an increased mesh density.



Model courtesy of Mark Hechel of MasterGraphics, Inc.

4. Save the drawing with the name `Completed15-2.dwg` to the Chapter 15 folder in the My Practice Drawings folder under My Documents and close the drawing.

### Exercise 15-3: Adding Sphere Primitives to the Bearing

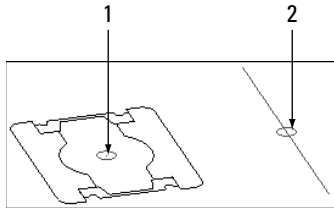
For this exercise, open the drawing file `Exercise15-3.dwg` from the Chapter 15 folder in the My Practice Drawings folder under My Documents. Then practice adding a series of 3D mesh primitive spheres to complete the 3D model of the bearing.

1. Turn off the layers `Bearing_Housing_Inner` and `Bearing_Housing_Outer` and set the layer `Bearings` current.



2. Start the 3D command and select the Sphere option.  
At the command prompt, enter **3d** and press Enter. Then enter **S** and press Enter.
3. At the Specify center point of sphere: prompt, select the point object indicated by 1 in Figure 15-5 using the Node object snap.

**Figure 15-5:**  
A 2D  
profile used  
to create  
the balls  
inside the  
bearing.



Model courtesy of Mark Hechel of MasterGraphics, Inc.

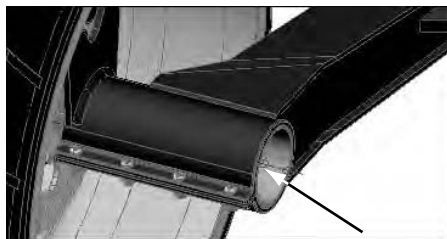
4. Use the Diameter option with a value of 0.49 for the sphere, and use 16 segments for both the longitudinal and latitudinal directions.
5. Choose Modify↔3D Operations↔3D Array.
6. When prompted to select objects, select the sphere that you just created and press Enter to end object selection.
7. Use the Polar array type, use 10 for the number of items in the array, and use an angle to fill of 360 degrees. When asked to rotate the objects, select either yes or no (because no matter how the sphere is rotated, it will be the same).
8. For the center point of the array, select the point object with the Node object snap indicated by 2 in Figure 15-5. Enable ortho mode and drag in the drawing along the Y axis and pick to define the axis for the array.



9. Set the base for the part to the point object that was indicated by 2 in Figure 15-5.  
At the command prompt, enter **base** and press Enter. Pick the point object using the Node object snap.
10. Turn on layers Bearing\_Housing\_Inner and Bearing\_Housing\_Outer; then freeze the layers 3Dconst and Profile.
11. Save the drawing with the name *Completed15-3.dwg* to the Chapter 15 folder in the My Practice Drawings folder under My Documents and close the drawing.
12. Open the drawing *RearAssembly.dwg* from the Chapter 15 folder in the My Practice Drawings folder under My Documents and attach the drawing *Completed15-3.dwg* on the layer Bearing at the point indicated by the arrow in Figure 15-6. As you attach the model of the bearing, you have to rotate it 90 degrees, or you can rotate it after it has been attached.

For information on attaching a drawing, see the exercises in Chapters 7 and 11.

**Figure 15-6:**  
Attaching  
the bearing  
to the rear  
assembly of  
the custom  
motorcycle.

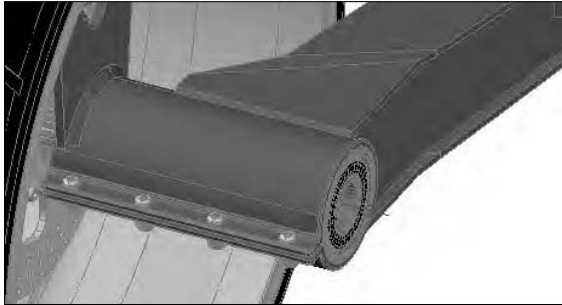


Model courtesy of Mark Hechel of MasterGraphics, Inc.

13. Save the drawing with the name `Completed-RearAssembly.dwg`. Close the drawing when you're done.

The model looks similar to Figure 15-7 when the part is attached.

**Figure 15-7:**  
The finished  
assembly  
with bearing  
attached.



*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

## Modeling with Surfaces

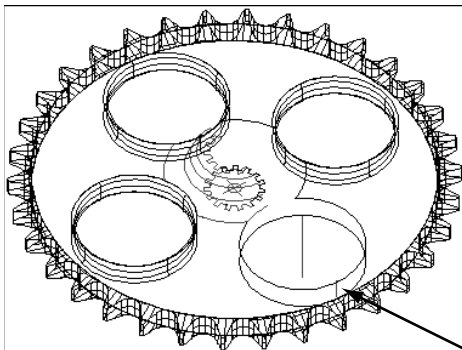
In this series of exercises, you finish a surface model of the rear gear for the custom motorcycle. You create extruded, lofted, and revolved surfaces and finish up parts of the model by mirroring certain elements.

### Exercise 15-4: Finishing the Holes in the Gear Using Revolved Surfaces

For this exercise, open the drawing file `Exercise15-4.dwg` from the Chapter 15 folder in the My Practice Drawings folder under My Documents. Then practice revolving a 2D profile around an axis. After the surface object is created, you change the number of isolines that are displayed.

1. Choose `Draw` → `Modeling` → `Revolve`.
2. When prompted to select the objects to revolve, select the 2D polyline indicated by the arrow in Figure 15-8 and press `Enter` to end selecting objects.

**Figure 15-8:**  
Preparing to  
revolve a 2D  
profile to  
create a  
surface  
object.



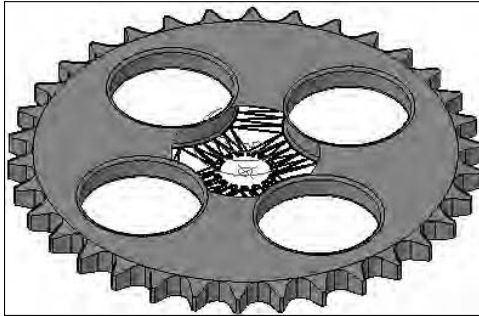
*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

3. Press `Enter` to use the default `Object` option to define the axis to be used for the revolution of the 2D polyline.
4. Select the line just to the left and up from the 2D polyline that you selected.
5. Press `Enter` to revolve the 2D polyline a full 360 degrees.

6. Erase the line that you used as the axis to revolve the 2D polyline.
7. Select the new surface object that was created and display the Properties palette.
8. In the Properties palette, set the U Isolines property to a value of 4 and the V Isolines property to a value of 1.
9. Apply the Conceptual visual style to the 3D model.

The drawing looks similar to the one in Figure 15-9.

**Figure 15-9:**  
A completed revolved surface.



Model courtesy of Mark Hechel of MasterGraphics, Inc.

10. Save the drawing with the name `Completed15-4.dwg` to the Chapter 15 folder in the My Practice Drawings folder under My Documents and close the drawing.

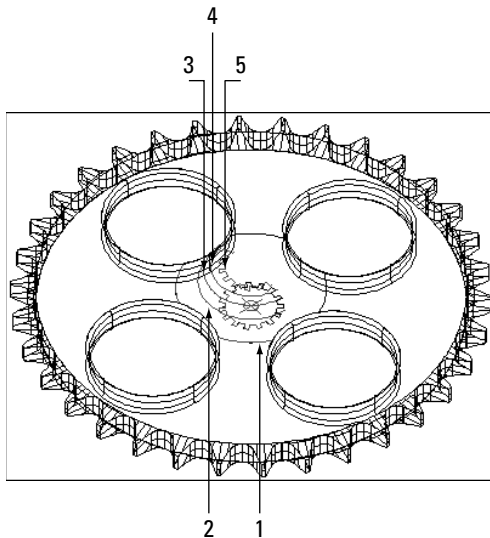


### Exercise 15-5: Creating a Hub for the Gear Using a Lofted Surface

For this exercise, open the drawing file `Exercise15-5.dwg` from the Chapter 15 folder in the My Practice Drawings folder under My Documents. Then practice creating a lofted surface object that will represent half of the hub for the gear.

1. Choose `Draw` → `Modeling` → `Loft`.
2. When prompted to select the cross sections for the lofted surface, select the objects indicated in Figure 15-10 from 1 to 5 and press `Enter` to end selecting objects.

**Figure 15-10:**  
Using cross sections to create a lofted surface.



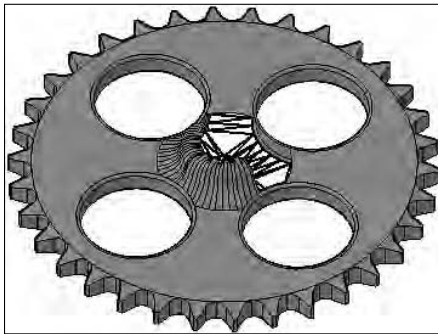
Model courtesy of Mark Hechel of MasterGraphics, Inc.



3. Press Enter to choose the default option of cross sections only when prompted for a lofting option.
4. In the Loft Settings dialog box, select the Smooth Fit radio button and then click OK.
5. Use the Properties palette to set both the U Isolines and V Isolines properties to a value of 1.
6. Apply the Conceptual visual style to the 3D model.

The drawing looks similar to the one in Figure 15-11.

**Figure 15-11:**  
Half of the  
hub for the  
gear cre-  
ated with  
a lofted  
surface.



Model courtesy of Mark Hechel of MasterGraphics, Inc.

7. Save the drawing with the name `Completed15-5.dwg` to the Chapter 15 folder in the My Practice Drawings folder under My Documents and close the drawing.

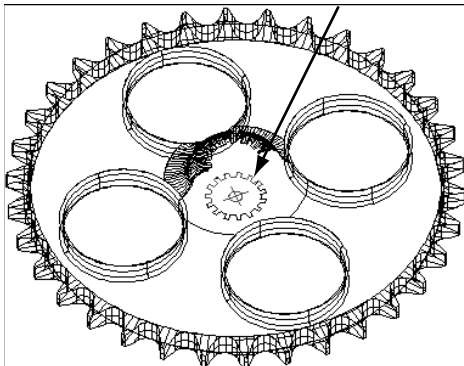
### Exercise 15-6: Completing the Center of the Gear Using an Extruded Surface



For this exercise, open the drawing file `Exercise15-6.dwg` from the Chapter 15 folder in the My Practice Drawings folder under My Documents. Then practice extruding a 2D profile to create an extruded surface, which completes half of the hub for the gear. After half of the hub is created, you mirror the two surfaces to complete the model and then attach it to the assembly.

1. Choose `Draw` → `Modeling` → `Extrude`.
2. When prompted to select the objects to extrude, select the 2D polyline indicated by the arrow in Figure 15-12 and press Enter to end selecting objects.

**Figure 15-12:**  
Extruding a  
2D polyline  
to complete  
one-half the  
center of  
the gear.



Model courtesy of Mark Hechel of MasterGraphics, Inc.

3. Drag the crosshairs up. Then enter **1** and press Enter to specify the height of the extruded surface.
4. Use the Properties palette to set both the U Isolines and V Isolines properties to a value of 1 for the newly extruded surface.
5. Use the MIRROR command and select the lofted and extruded surfaces that are located in the center of the gear when prompted to select objects.

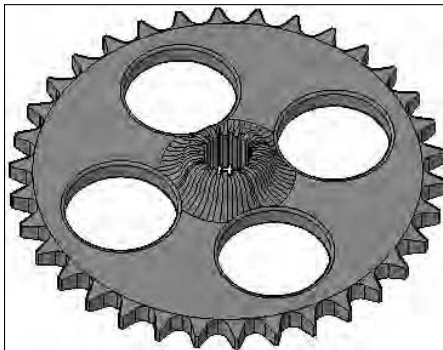
The lofted surface is located above the newly created extruded surface. It's the same surface that you created in Exercise 15-5.

6. When prompted for the mirror line, use the Node object snap and select the point object at the center of the gear. Turn on ortho mode and drag to the right of the model and pick. Make sure you keep the original objects.

Both surface objects are mirrored, which completes the center of the hub and the 3D model of the gear.

7. Apply the Conceptual visual style to the 3D model.

The drawing looks similar to the one in Figure 15-13.



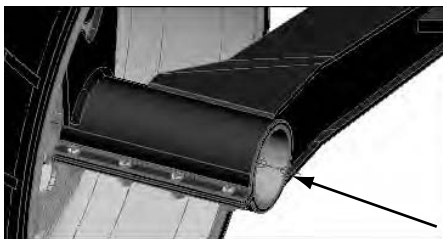
**Figure 15-13:**  
A completed gear model using surfaces.

*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

8. Save the drawing with the name `Completed15-6.dwg` to the Chapter 15 folder in the My Practice Drawings folder under My Documents and close the drawing.
9. Open the drawing `Completed-RearAssembly.dwg` if you completed Exercise 15-3 or `RearAssembly.dwg` from the Chapter 15 folder in the My Practice Drawings folder under My Documents. Attach the drawing `Completed15-6.dwg` on the layer `Chain_gear` at the point indicated by the arrow in Figure 15-14.

For information on attaching a drawing, see the exercises in Chapters 7 and 11.

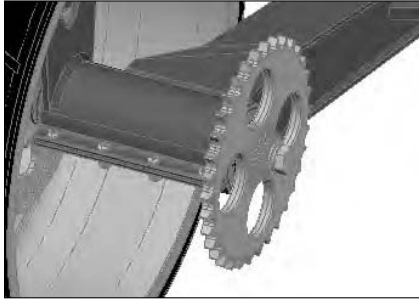
**Figure 15-14:**  
Attaching the gear to the rear assembly of the custom motorcycle.



*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

10. After you attach the part, rotate it 90 degrees along the Y axis.
11. Turn on the layers Axel and Axel\_fastners to see the objects that are used to attach the gear to the assembly.
12. Save the drawing with the name `Completed-RearAssembly.dwg`. Close the drawing when you're done.

The model looks similar to Figure 15-15 when the part is attached.



**Figure 15-15:**  
A completed assembly.

*Model courtesy of Mark Hechel of MasterGraphics, Inc*



# Part V

## Visualizing the Design

The 5<sup>th</sup> Wave

By Rich Tennant



"I've never seen a 3D modeling program do that before."

## *In this part . . .*

**L**ighting and materials bring your 3D models to life by giving them realistic colors and textures and providing depth through lit and shaded areas. You can add lighting to emulate small light sources like LEDs and much larger sources of lighting like the sun. Materials give your models a realistic look through textures, patterns, and colors.

You can then generate a rendering of the model with the materials applied and have AutoCAD calculate how the lights in a drawing emit light and cast shadows. Along with discovering how to render still images that you can save to a file, you find out how to create a basic animation of your model that you can share with others.

# Chapter 16

# Lighting

---

## *In This Chapter*

- ▶ Shedding some light on lighting
  - ▶ Creating user-defined lights
  - ▶ Modifying lights
  - ▶ Adjusting global lighting
- 

**L**ighting in AutoCAD allows you to re-create light sources in your 3D models to do one of two different things. It adds realistic lighting and highlights to your models that would be present in a manufactured part such as an LED (light-emitting diode) or a ceiling light hanging overhead of your assembly of a large machine. You can also create artificial lighting that might not be present in the real world but that helps you draw focus to a specific part or feature in your 3D model.

But highlighting can also make a 3D model look artificial and unbelievable if not done properly. The exercises in this chapter show you how to create and modify some of the different types of lights and lighting systems that you can use with your 3D models.

## *Overview of Lighting*

Lighting in AutoCAD is composed of two types of lighting systems: default and user-defined. The default lighting system is present in each drawing that you create. The user-defined lighting system requires a bit more work than just opening a drawing, as well it should because you're setting it up. Starting with AutoCAD 2008, user-defined lighting comes in two different types: standard and photometric. Standard lighting was available in AutoCAD 2007 and prior releases. Photometric lighting is new in AutoCAD, but it has been around for a number of years in other software packages, like 3ds Max and Autodesk Viz.

To use user-defined light or Sunlight system, you must turn off DEFAULTLIGHTING.



TIP

When rendering a 3D model that contains viewports with both default lighting and user-defined lighting, you can force user-defined lights for all viewports with the system variable RENDERUSERLIGHTS set to a value of 1. If you want a combination of default lighting and user-defined lights for different floating viewports in your drawing, you need to set RENDERUSERLIGHTS to a value of 0. The RENDERUSERLIGHTS system variable makes it much easier to enable the use of user-defined lights instead of setting DEFAULTLIGHTING to 0 for each floating viewport in your drawing.



WARNING!

When using user-defined lights and creating renderings, the results that you get may vary from those shown in this book. The reason for the differences is due to a number of factors, such as monitor and video card settings. The brightness and contrast of the monitor can make the images appear too dark or washed out. So you might have to work with the intensity settings of the lights used in the exercises to get the best results. When sharing renderings electronically, ask your co-workers whether the renderings created might be too dark or light. Often, printing a rendered image gives you the best control over the final product.

## Default lighting

Default lighting isn't a complex lighting system to use — especially because AutoCAD does all the work for you — but it isn't useful when trying to create realistic renderings that have shadows, and it doesn't support attenuation like user-defined lights do. Default lighting turns out quick and basic renderings that don't have shadows. As you change the view of your drawing with default lighting enabled, the default lights move with the current view so they aren't stationary, like user-defined lights

The system variable `DEFAULTLIGHTING` determines whether default lighting is used during rendering. By default, `DEFAULTLIGHTING` is set to a value of 1, indicating that default lighting is enabled. You can disable `DEFAULTLIGHTING` by setting it to a value of 0.

When you use default lighting, you have a choice of using one or two lights; by default, two lights are used. To change the number of lights, use the system variable `DEFAULTLIGHTINGTYPE`. When the system variable `DEFAULTLIGHTINGTYPE` is set to 1, two over-the-shoulder lights are used, whereas the `DEFAULTLIGHTINGTYPE` set to 0 (zero) uses one light. The values assigned to `DEFAULTLIGHTING` and `DEFAULTLIGHTINGTYPE` are saved with the drawing and can be set in a custom drawing template that you use to create your new drawings from.

## User-defined lighting

User-defined lights require more work on your part than using the default lighting system, but the rewards are worth the time it takes to get it right.

For each viewport that you want to use user-defined lights in, set `DEFAULTLIGHTING` appropriately with the following methods:

- ✓ Enter **defaultlighting** at the command prompt, and when prompted, enter a value of **0** to disable default lighting or **1** to enable default lighting.
- ✓ Click the Viewport Lighting Mode toggle on the Lights control panel on the Dashboard palette to toggle between the use of default lighting and user-defined lights in the current viewport.

When you create user-defined lights, you can use either standard or photometric lighting:

- ✓ **Standard:** This type of lighting was in all AutoCAD versions up to AutoCAD 2007. When `LIGHTINGUNITS` is set to 0, standard lights are used. You can create three types of user-defined lights — point, spotlight, and distant lights — along with using the Sunlight system.

When you use standard lights, you have to deal with certain restrictions on which types of lights are available — and in some cases certain settings for lighting and rendering. The Sunlight system is restricted from simulating the sky, the Sun & Sky background option isn't available for named views or cameras, rendering exposure isn't supported, and midtones aren't available for global lighting. Outside those limitations, you can use standard lights to add lighting to your 3D models, which can add depth to your renderings by highlighting specific parts and cast shadows.



✓ **Photometric:** This type of light is new to AutoCAD 2008. Photometric lights are used during rendering when LIGHTINGUNITS is set to a value of 1 or 2. A value of 1 indicates that International lighting units are used, and a value of 2 indicates that American lighting units are used. Photometric lights represent lights in the real world. They're similar to standard lights, with the exception that photometric lights always attenuate and define the intensity of a light. In addition to being able to define lights of the types point, spotlight, and distant and to use the Sunlight system, you can create weblights.

Photometric lights are the preferred type of lights because they offer many more options over standard lights, and they do a better job of representing lights in the real world. The down side to photometric lights is that they don't offer the ability to control the attenuation values of the light like standard lights do.

You can change the lighting units. Open the Drawing Units dialog box (Format↔Units) and select one of the available options from the Units for Specifying the Intensity of Lighting drop-down list under the Lighting section. From the Units for Specifying the Intensity of Lighting drop-down list, you can choose Generic, International, or American. Generic allows you to use standard lights, while International and American are used to enable the use of photometric lights.

## Lights in blocks

Referencing drawings is very common when creating complex 3D models, but when you reference or insert a drawing that contains user-defined lights, they're brought into the drawing as well. You have the option to allow user-defined lights created as part of a block or reference drawing to participate in emitting light during rendering. The system variable LIGHTSINBLOCK controls whether lights contained in blocks affect the lighting of a 3D model. If LIGHTSINBLOCK is set to a value of 1, lights contained in blocks are used during rendering; otherwise, a value of 0 disables lights contained in blocks, and they're ignored during rendering.



1. In the exercises in the chapter, you add lights to a model. Before doing so, you should render your drawing to see how it realistically looks before adding lights. To render a drawing, choose View↔\_\_\_\_\_↔\_\_\_\_\_.

## Creating User-Defined Lights

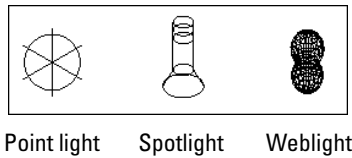
User-defined lights give your renderings greater depth and visual appeal than default lighting because you control the color of the light that is emitted, the direction the light is emitted in, and the intensity in which the light is emitted. AutoCAD allows you to create four different user-defined lights in your drawings: point lights, spotlights, weblights, and distant lights. You can also take control of the Sunlight system, which allows you to adjust the angle and location of the Sun in a drawing.

After you add a light to the drawing, a graphical representation known as a *glyph* is displayed. Figure 16-1 shows the three different types of light glyphs. The glyph for a point light looks like a circle with lines running through it, the glyph for a spotlight looks like a flashlight, and the glyph for a weblight looks like a mesh object. The glyph for a weblight looks different based on the IES file that's assigned to it.



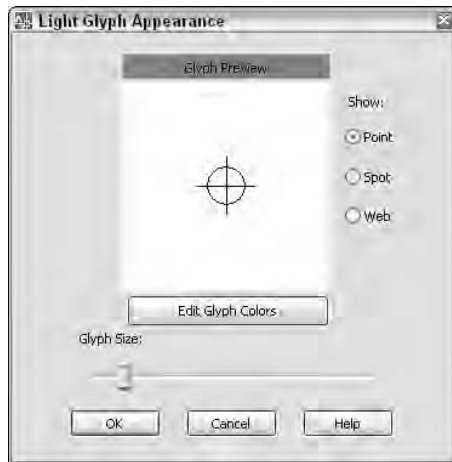
Distant lights and the Sunlight system don't have glyphs displayed for them in the drawing window to designate where they're located or the direction in which they emit light.

**Figure 16-1:**  
Light glyphs.  
From left to  
right: point  
light, spot-  
light, and  
weblight.



If you don't see light glyphs in the drawing, click the Light Glyphs toggle in the expanded section of the Lights control panel on the Dashboard palette or change the value of the system variable LIGHTGLYPHDISPLAY to 1. Light glyphs by default don't plot unless the Plot Glyph property for the light is set to Yes in the Properties palette.

You can control the size of the light glyphs by clicking the Light Glyph Settings button on the Drafting tab of the Options dialog box to display the Light Glyph Appearance dialog box (see Figure 16-2). In the Light Glyph Appearance dialog box, change the size of the light glyphs and their color.



**Figure 16-2:**  
The Light  
Glyph  
Appearance  
dialog box.

## Point lights

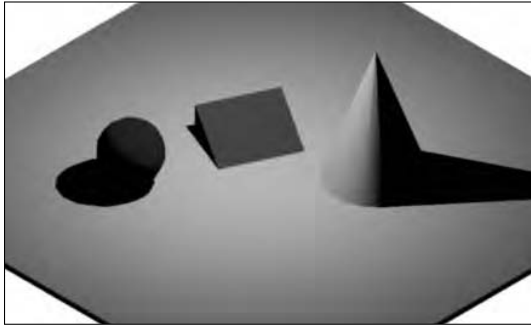
Point lights are one of the most commonly used user-defined lights in AutoCAD. A *point light* is similar to a candle or a light bulb because it emits light equally in all directions. Although point lights do emit light in all directions, you can define a point light with a target in mind, which doesn't affect how the light is emitted.

You create point lights two ways:

- ✓ Use the LIGHT command, with either the Point or Targetpoint option.
- ✓ Choose View⇨Render⇨Light⇨New Point Light or click Create a Point Light on the Lights control panel.

Figure 16-3 shows a point light over a wedge and between a sphere and cone primitive. The light is emitted equally away from the center of the point light in all directions.

**Figure 16-3:**  
A point light  
used to add  
lighting to a  
grouping of  
3D solids.



## Spotlights

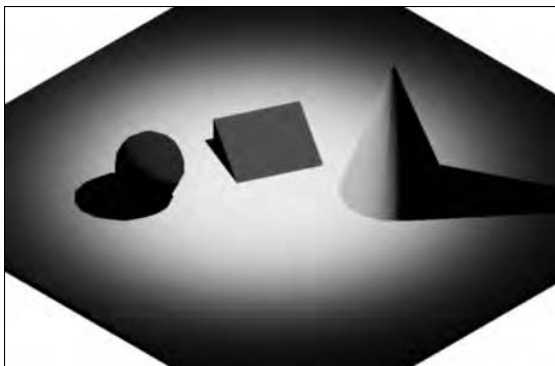
Spotlights are the second most commonly used user-defined lights in AutoCAD. A *spotlight* is similar to a flashlight or a movie projector because it emits light in a cone shape and in a specific direction. Spotlights use hotspot and falloff properties to define the brightest light emitted along the path and where the light starts to dim. The innermost part of the cone of light is known as the *hotspot*, and it is the brightest part of the light that's emitted; the area outside the hotspot is where the light becomes dimmer the farther away it gets from the hotspot. Like point lights, you can define spotlights with or without a target.

You create spotlights with the following methods:

- ✓ Use the LIGHT command, with either the Spot or Freespot option.
- ✓ Choose View⇨Render⇨Light⇨Create Spotlight.
- ✓ Click Create a Spotlight on the Lights control panel.

Figure 16-4 shows a spotlight over a wedge and between a sphere and cone primitive. Notice that the light is the brightest over the wedge and gets dimmer the farther it moves away.

**Figure 16-4:**  
A spotlight  
adds light-  
ing to a  
grouping of  
3D solids.



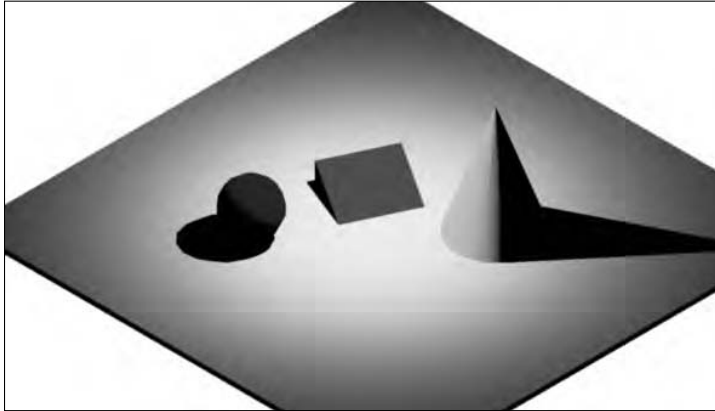
## Webights

Webights are the newest type of light in AutoCAD and are available only with photometric lighting. A *webight* is a cross between a point light and a spotlight due to the way it emits light. Instead of emitting light along a path in one direction or in all directions, a webight emits light by the IES file it is assigned. An IES (Illuminating Engineering Society) file contains information about how an actual light in the real world emits light. Manufacturers of light fixtures provide IES files, and you can often obtain them for free from the manufacturer's Web site.



AutoCAD ships with example IES files located in the Webfiles folder. To locate the Webfiles folder on your computer, open the Options dialog box and click the Files tab. On the Files tab, expand the Web File Search Path node.

You can create weblights with the LIGHT command and either the Web or Freeweb option. Figure 16-5 shows a weblight over a wedge and between a sphere and cone primitive. The light is the brightest closest to the wedge and gets dimmer the farther it moves away.



**Figure 16-5:** Weblight adds lighting to a grouping of 3D solids.

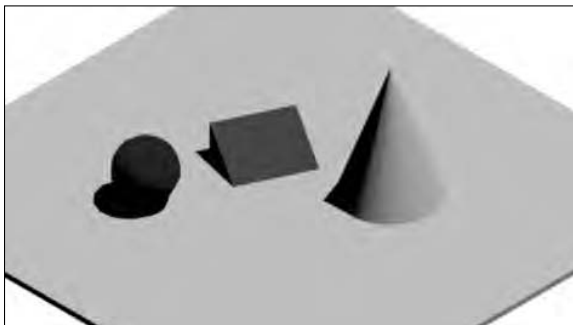
## Distant lights

Distant lights fill a drawing with a light that has a consistent color and intensity. A *distant light* is like the Sun because it emits light in one direction and doesn't attenuate (fall off) like other user-defined lights do. When you create a distant light, you specify the direction in which the light is emitted.

You create distant lights with the following methods:

- ✓ Use the LIGHT command with the Distant option.
- ✓ Choose View⇨Render⇨Light⇨New Distant Light.
- ✓ Click Create a Distant Light on the Lights control panel.

Figure 16-6 shows a distant light brightening a drawing containing wedge, sphere, and cone primitives. The distant light in the drawing emits light at an angle, and the light emitted doesn't increase or decrease in intensity like the other lights do.



**Figure 16-6:** Distant light adds lighting to a grouping of 3D solids.



### Exercise 16-1: Creating User-Defined Lights

For this exercise, open the drawing file `Exercise16-1.dwg` from the Chapter 16 folder in the My Practice Drawings folder under My Documents. Then practice creating a point light and distant light. You also create a basic rendering of the drawing to see what the lights look like with and without user-defined lights.

1. Turn off Dynamic input.
2. Choose `View↔Render↔Render`.
3. Choose `Format↔Units`.
4. In the Drawing Units dialog box, select American from the Units for Specifying the Intensity of Lighting drop-down list under the Lighting section. Click OK to exit the dialog box.
5. Choose `View↔Render↔Light↔New Point Light`.
6. When the Viewport Lighting Mode message box is displayed, click Yes to disable default lighting mode.
7. At the Specify source location `<0,0,0>`: prompt, enter **86,76,6** and press Enter.
8. At the Enter an option to change [Name/Intensity factor/Status/Photometry/shadow/Attenuation/filterColor/eXit] `<eXit>`: prompt, enter **N** and press Enter.
9. At the Enter light name `<Pointlight1>`: prompt, enter **accent-light** and press Enter twice.
10. Render the model again to see what effect the point light had on the 3D model.

The rendering looks similar to Figure 16-7. The model appears washed out now, and a shadow is cast behind the part.

**Figure 16-7:**  
The results of the point light in the 3D model.



11. At the command prompt, enter **light** and press Enter.
12. At the Enter light type [Point/Spot/Web/Targetpoint/Freespot/freeweB/Distant] `<Point>`: prompt, enter **D** and press Enter.
13. When the Photometric Distant Light message box is displayed, click OK to continue to place the distant light.
14. At the Specify light direction FROM `<0,0,0>` or [Vector]: prompt, enter **92,75,5** and press Enter.
15. At the Specify light direction TO `<1,1,1>`: prompt, enter **89,80,0.5** and press Enter.
16. At the Enter an option to change Enter an option to change [Name/Intensity factor/Status/Photometry/shadow/filterColor/eXit] `<eXit>`: prompt, enter **N** and press Enter.

17. At the `Enter light name <Distantlight2>`: prompt, enter **fill-light** and press Enter twice.
18. Render the model again to see what effect the distant and point light have on the 3D model.

The rendering looks similar to Figure 16-8. The model appears even more washed out, and the shadow cast by the point light appears lighter. You fix the washed-out appearance in Exercise 16-2.

**Figure 16-8:** The results of the distant light in the 3D model.



19. Save the drawing with the name `Completed16-1.dwg` to the Chapter 16 folder in the My Practice Drawings folder under My Documents and close the drawing.



2. Exercise 16-1 shows how to add point lights and spotlights to a model. What steps do you take to create a distant light?

1. \_\_\_\_\_
2. \_\_\_\_\_
3. \_\_\_\_\_
4. \_\_\_\_\_
5. \_\_\_\_\_

For more information on using user-defined lights, see the topic *Add Lighting to Your Model* in the AutoCAD help system. To locate the topic, display the Online Help system and click the Contents tab. On the Contents tab, choose *User's Guide* → *Create Realistic Images and Graphics*.

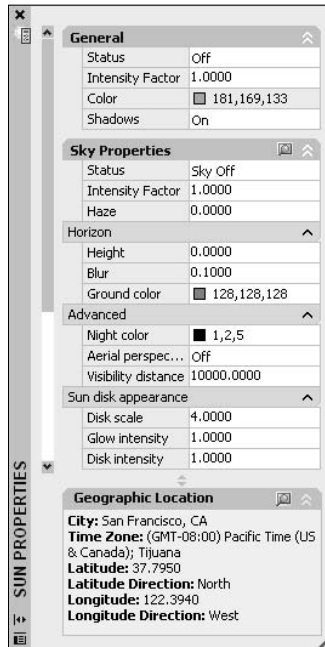
## Sunlight

The Sunlight system in AutoCAD allows you to replicate the light emitted from the Sun. It is usually used for exterior renderings of buildings or interior renderings where light might come in through a window. You specify the location and angle of the Sun relative to a specific location on the planet defined by longitude, latitude, and the direction of North. You enable the Sunlight system by clicking the Sun Status button on the Lights control panel or setting the Status property in the Sun Properties palette to On. The Sun Properties palette is where you do most of the configuring for the Sunlight system, but you can also use the Date and Time sliders on the Lights control panel.

To display the Sun Properties palette, click the Edit the Sun button under the expanded section of the Lights control panel or choose *View* → *Render* → *Light* → *Sun Properties*. With the Sun Properties palette (see Figure 16-9), you can specify the calendar day, time

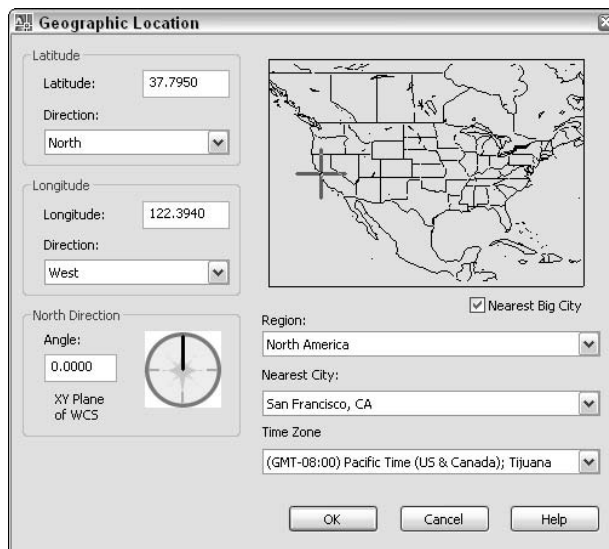
of day, and various other properties that affect the positioning and color of light emitted by the Sun. When you use photometric lighting, you can simulate the sky as the background of the current view and display a disc in the sky to show the location and intensity of the Sun.

**Figure 16-9:**  
The Sun Properties palette allows you to configure the Sunlight system.



The Geographic Location dialog box specifies the location on the planet in which the 3D model resides (see Figure 16-10). By using a geographic location, you accurately represent the location of the Sun and simulate the sky at specific time intervals. To display the Geographic Location dialog box, click the Launch Geographic Location button in the Geographic Location category or click Geographic Location in the expanded section of the Lights control panel.

**Figure 16-10:**  
The Geographic Location dialog box determines the position of the Sun based on a location on the planet.

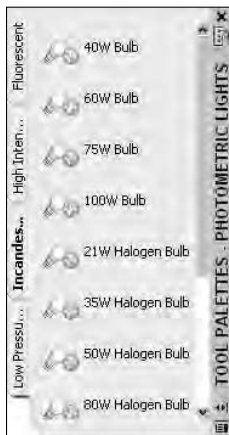


For more information on using the Sunlight system, see the topic Sun and Sky Simulation in the AutoCAD help system. To locate the topic, display the Online Help system and click the Contents tab. On the Contents tab, choose User's Guide→Create Realistic Images and Graphics→Add Lighting to Your Model→Sun and Sky Simulation.

## Using Predefined Lights from Tool Palettes

AutoCAD comes installed with several tool palettes of predefined lights. You can choose between generic (or standard) lights and photometric lights. You can access the predefined lights from the Generic Lights and Photometric Lights tool palette groups on the Tool Palettes Window (see Figure 16-11). To access these tool palette groups, display the Tool Palettes Window (choose Tools→Palettes→Tool Palettes) and right-click the title bar of the Tool Palettes Window when it isn't docked, or right-click the double grip bar at the top when it is docked. From the shortcut menu, choose either Generic Lights or Photometric Lights. Then click one of the tools to place a light based on the predefined settings or click a tab to switch between the available tool palettes in the selected group.

**Figure 16-11:** The Tool Palettes Window with the Photometric Lights tool palette group displayed.



## Modifying Lights

Lights are like other objects that you find in a drawing; they have properties that you can modify at any time to adjust the way they affect your 3D model during rendering.

Table 16-1 lists the common properties that user-defined lights have.

<i>Property Name</i>	<i>Description</i>
Name	The name assigned to a user-defined light. AutoCAD assigns a default name for each light that you create. For example, if you create a point light, AutoCAD assigns it the name Pointlight1. For each new light that you create, AutoCAD increments the number at the end of the name by 1.
Intensity/ Intensity Factor	Specifies the brightness level of the light; the greater the intensity value, the brighter the light emitted.



<i>Property Name</i>	<i>Description</i>
Status	Controls whether the light is currently enabled or disabled. When the light is disabled, it isn't used during rendering.
Photometry	The photometry property is available only when photometric lighting is enabled. This option allows you to specify the intensity of a photometric light by using the measurement values of Candela, Lux, or Foot-candle. Along with the intensity, you can choose one of the standard color names or a Kelvin temperature for the light, which affects the color of light emitted.
Shadow	Controls whether a light casts shadows and which type of shadow is cast. You can choose among Sharp, Soft Mapped, and Soft Sampled shadows. Sharp shadows are the fastest to be created during rendering, but they don't look as nice as Soft Mapped or Soft Sampled shadows.
Attenuation	Determines whether the emitted light falls off and at what distance the falloff ends. All lights with the exception of the Sunlight and distant lights can have attenuation, which affects how light decreases the farther it gets from the source. You can choose to use no attenuation, or you can use either Inverse Linear or Inverse Square attenuation. The Inverse Square attenuation option emulates the way lights in the real world fall off and is the only option when you use photometric lighting.
Color/Filter Color	Controls the color of the light emitted. You could use a green color for an LED or a soft yellow color for an incandescent light bulb.

When adjusting a light using grips, you can adjust the following:

- ✓ The source location of the light and its target point (if it has one).
- ✓ The hotspot and falloff of a spotlight.
- ✓ The attenuation limits of a standard light. (You can adjust this right in the drawing window as well.)

To adjust a light by using grips, select the glyph and then the grip that you want to adjust.



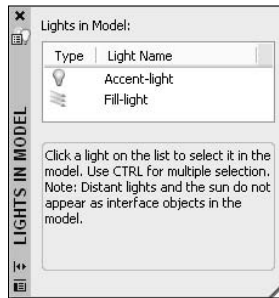
3. To adjust lights using grips, you must have grip editing enabled. How do you enable grip editing?
- A. In the Options dialog box
  - B. On the Properties palette
  - C. In the Tool Palettes Window

If you want to adjust properties of a light other than the source point, target point, attenuation limits, hotspot, and falloff, use the Properties palette. From the Properties palette, you can adjust the following:

- ✓ Turn a light on or off
- ✓ Control the type of shadows it casts, intensity level, and other settings

If you can't see a light in the drawing to edit its properties or you have light glyphs turned off, you can use the Lights in Model palette (see Figure 16-12) to select a light in the drawing file. The Lights in Model palette is the only way to select a distant light to edit its properties. You can then right-click and select Properties to display the Properties palette or select Delete Light to remove the light from the drawing.

**Figure 16-12:**  
Lights in  
Model  
palette.



The Lights in Model palette displays the name of each light in the drawing, and one of four icons indicates what type of light you're selecting. To display the Lights in Model palette, click Light List on the Lights control panel or choose Tools⇨Palettes⇨Lights.



### Exercise 16-2: Modifying User-Defined Lights

For this exercise, open the drawing file `Exercise16-2.dwg` from the Chapter 16 folder in the My Practice Drawings folder under My Documents. Then practice modifying lights.

1. Choose Tools⇨Palettes⇨Lights.
2. On the Lights in Model palette, select the Accent-light option and right-click. From the shortcut menu, select Properties.
3. On the Properties palette, set the property Intensity Factor to a value of 0.05.
4. Click the Filter Color property and select Select Color from the drop-down list. In the Select Color dialog box, click the Index Color tab and select color 61. Click OK to exit the Select Color dialog box.
5. On the Lights in Model palette, select the Fill-light and set the property Intensity Factor to a value of 0.5 on the Properties palette
6. Click the Shadows property and select Off from the drop-down list.
7. Render the model to see what effect changing the properties of the lights had on the 3D model.

The rendering looks similar to Figure 16-13. The model appears much better (compare it to Figure 16-8) and the specific features of the 3D model are clear now.

**Figure 16-13:**  
The results  
of the  
changes  
made to the  
lights in the  
3D model.



8. Save the drawing with the name `Completed16-2.dwg` to the Chapter 16 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Global Lighting

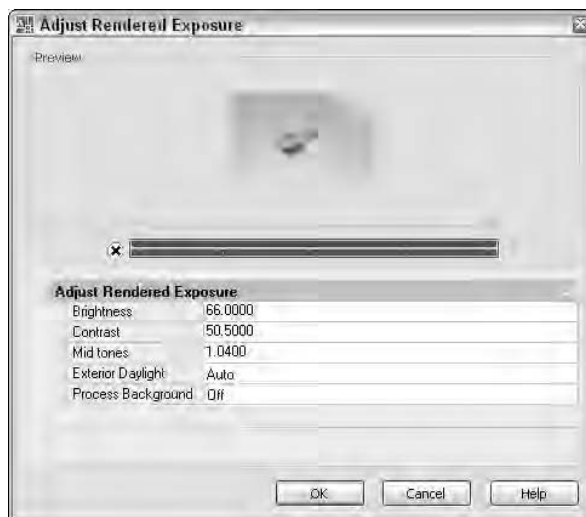
Global lighting affects a rendering no matter what kind of lighting you use. To adjust the global lighting, click the double down arrows located on the Lights control panel to expand the panel (see Figure 16-14). You can change the brightness and contrast levels globally for the entire rendering. Doing so allows you to increase the brightness level of a rendering if it appears too dark overall or decrease the brightness level if it appears too washed out. In addition to brightness and contrast, you can adjust the level of midtone values that can help bring out additional details in a rendering. When LIGHTINGUNITS is set to 0, the midtones slider isn't accessible but is available when LIGHTINGUNITS is set to 1 or 2. Both the brightness and contrast sliders are accessible no matter what the current value of LIGHTINGUNITS is set to.

**Figure 16-14:**

The global lighting controls on the Lights control panel.



If you have LIGHTINGUNITS set to a value of 1 or 2, you can also adjust global lighting using the Adjust Rendered Exposure dialog box (see Figure 16-15). Using the Adjust Rendered Exposure dialog box gives you a mini-preview of what the most recent rendering would look like with the new exposure settings. You don't have to render the drawing first to make sure it looks like you want it to. You can display the Adjust Rendered Exposure dialog box by using the RENDEREXPOSURE command or by clicking the Adjust Exposure button in the expanded section of the Render control panel on the Dashboard.



**Figure 16-15:**  
The Adjust Rendered Exposure dialog box.



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### Exercise 16-3: Adjusting Global Lighting

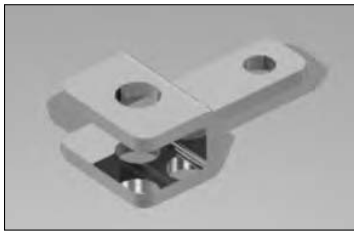
For this exercise, open the drawing file `Exercise16-3.dwg` from the Chapter 16 folder in the My Practice Drawings folder under My Documents. Then practice adjusting the brightness, contrast, and midtones globally for the drawing.

1. Display the Dashboard palette and click the Render button on the Render control panel.  
A rendering of the drawing is created in the Render Window.
2. Expand the Lights control panel and slide the Brightness slider until a value near 50 appears in the text box to the right of the slider.
3. Slide the Contrast slider until a value near 45 appears in the text box to the right of the slider.
4. Create another rendering of the drawing to see what the changes look like.  
You can compare the current rendering with the previous rendering by clicking the previous output file at the bottom of the Render Window. The most recent rendered file is always at the top of the list.
5. Slide the midtone slider on the Lights control panel until a value near 2.5 appears in the text box to the right of the slider.
6. Create another rendering of the drawing and compare the new rendering with the previous rendering.

The final rendering of the drawing looks similar to Figure 16-16.

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**Figure 16-16:**  
Rendering with revised lighting by changing the global lighting for the model.



7. Save the drawing with the name `Completed16-3.dwg` to the Chapter 16 folder in the My Practice Drawings folder under My Documents and close the drawing.

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### Exercise 16-4: Using the Render Exposure Dialog Box

For this exercise, open the drawing file `Exercise16-3.dwg` from the Chapter 16 folder in the My Practice Drawings folder under My Documents. Then practice adjusting the brightness and contrast globally by using the Adjust Render Exposure dialog box.

Change some of the settings in the dialog box and see how they affect the Preview area. After making a few changes, create a rendering of the drawing. Save the drawing with the name `Completed16-4.dwg` to the Chapter 16 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Turning Old into New

Lights in AutoCAD 2008 are stored in drawing files in a different way than they were in AutoCAD 2006 and prior, so if you have a drawing that contains old lights and want to use them, you must use the CONVERTOLDLIGHTS command to change all the old lights in a drawing to the new type of lights. If you don't convert the old lights, they won't be used for rendering. If old lights are found and converted in the drawing, the number of lights that were converted is displayed in the AutoCAD Text Window. For example, the message `3 light(s) converted.` indicates that three old lights were converted in the drawing.

Remembering the CONVERTOLDLIGHTS command can be a task in itself, so you can have AutoCAD update old lights automatically for you by setting the system variable 3DCONVERSIONMODE to a value of 1 (which is the default value). 3DCONVERSIONMODE can also be set to 0, which disables the automatic conversion of old lights, or to a value of 2, which prompts you when old lights are available for converting.

## Answers to Practice Opportunities

**1. To render a drawing, choose View⇨Render⇨Render.**

Review the rendered image of the drawing in the Render Window.

**2. What steps do you take to create a distant light?**

1. Choose View⇨Render⇨Light⇨New Distant Light.
2. Disable default lighting if it is enabled.
3. Specify the FROM and TO directions for the distant light.
4. Use the Name option and enter a new name.
5. Use the other options that the command offers and press Enter to complete the command.

**3. How do you enable grip editing?**

A. In the Options dialog box (choose Tools⇨Options). Click the Selection tab, and select the Enable Grips check box under the Grips section.

For more information on enabling grip editing, see Chapter 9.



# Chapter 17

# Materials

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## *In This Chapter*

- ▶ Creating and modifying materials
  - ▶ Accessing predefined materials from libraries
  - ▶ Attaching materials to and removing materials from 3D objects
  - ▶ Mapping materials
  - ▶ Converting old materials to new materials
- 

**M**aterials are the key to making your 3D models look realistic; lighting alone can't make solid colors appear lifelike, nor is everything in the real world only defined by solid colors. Although materials are one of the most advanced features in AutoCAD, they're not complex. You can create your own materials from scratch or use predefined materials. AutoCAD comes with a large number of predefined materials that are organized into libraries. You can also create your own library of the materials from the ones that you create and use for your 3D models.

If you don't like how a specific material looks, you can adjust the properties, such as Diffuse Color, Opacity, and Self-Illumination. Each of the properties of a material helps you re-create a material found in the real world.

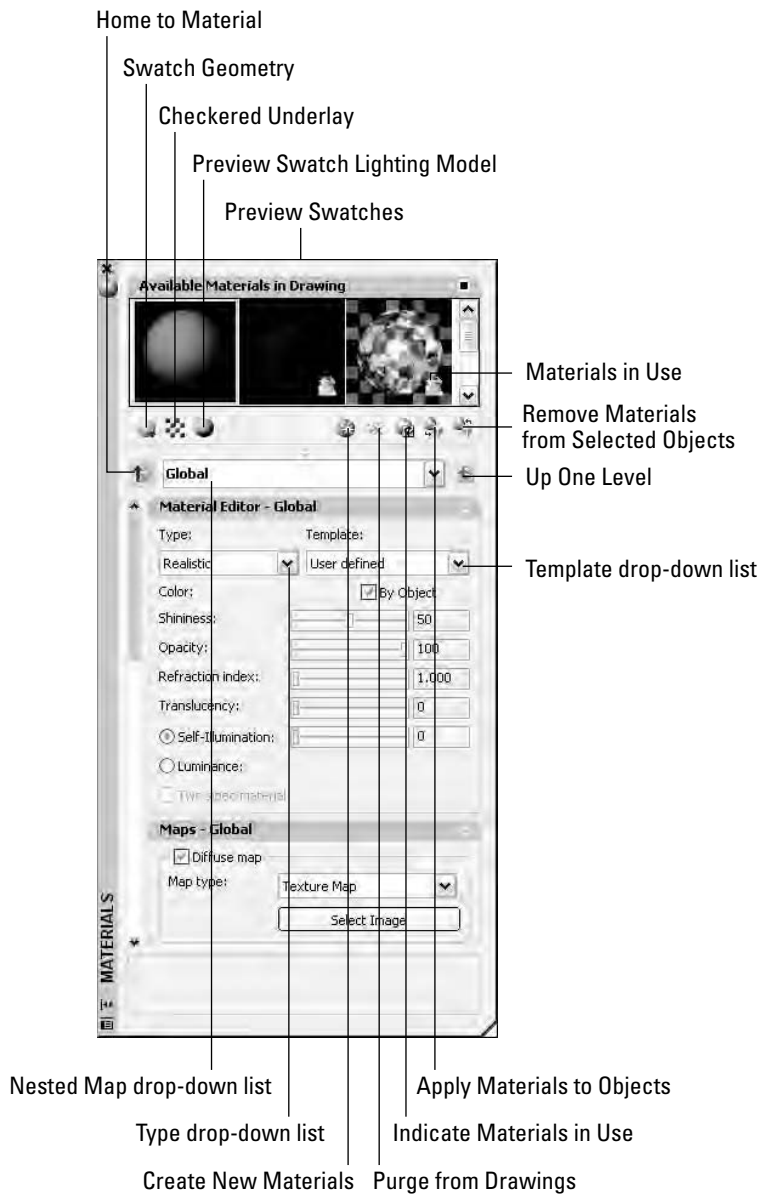
After you define a material, all you have to do is attached it to a 3D object or face. You can modify a material at any time, allowing you to tweak the color or other attributes of a material as needed. You can also adjust the direction in which a material is applied and where the pattern starts from through a process called *material mapping*. After you attach and map materials, you can then generate a rendering based on the lights and materials in the 3D model. You might need to adjust materials based on the lights in the 3D model or adjust a light to give the desired effect of an attached material.

The exercises in this chapter show you how to create, modify, and attach materials.

## *Creating and Modifying Materials*

You create and modify materials with the Materials palette (see Figure 17-1). The Materials palette is divided into different sections. It controls which materials are available in the drawing and the different properties:

- ✓ **Check which materials are currently defined.** The Available Materials in Drawing category allows you to see which materials are currently defined in the drawing and which ones are currently being used in the Preview Swatches area. Materials currently in use have a small drawing-file icon displayed in the lower-right corner of the preview swatch (see the center preview swatch in Figure 17-1). You can refresh the status of the materials in use in the drawing by clicking Indicate Materials in Use.



**Figure 17-1:**  
You can use the Materials palette to create and modify materials.

- ✔ **Switch the geometry of the material.** Choose to show a box or cylinder instead of a sphere from the Swatch Geometry drop-down menu.
- ✔ **Add an underlay.** Display a checkered background by using the Checkedered Underlay toggle.
- ✔ **Choose how many lights you want.** Specify whether you want one or two lights by choosing from the Preview Swatch Lighting Model drop-down menu.
- ✔ **Create new material.** Click the Create New Material button. The Create New Material dialog box opens, which allows you to give the material a name and optionally a description. AutoCAD by default assigns a default name of Material 1, Material 2, and so on.
- ✔ **Set the material to a specific type.** Select a type from Type drop-down list. The



types that you can select are Realistic, Realistic Metal, Advanced, and Advanced Metal.

- ✔ **Choose a template.** If you select either Realistic or Realistic Metal from the Type drop-down list, you can select one of the 22 material templates from the Templates drop-down list. Some of the templates that you can choose are Metal – Brushed, Plastic, Water, Mirror, and Glass – Clear.
- ✔ **Choose global properties.** The type of material that you select from the Type drop-down list affects which properties are available for the material under the Material Editor – Global category. For example, if you select Realistic, you can specify a value for Opacity and Translucency, but these same properties aren't available when you select Realistic Metal.
- ✔ **Define a map.** Maps allow you to use image files and what are known as *procedural maps* (defined under the Maps – Global category). Procedural maps are dynamically created images that allow you to create unique floor tiles, wood grains, and checker patterns, among a few others. When you use images and procedural maps, you can adjust the scaling and tiling of the pattern along with an offset value to control how the image tiles. Additionally, some material types offer Advanced Lighting Override properties that affect the way light interacts with the material.
- ✔ **Navigate materials.** Materials can have nested maps and images to represent floor tiles and wall textures. The Home to Material and Up One Level buttons allow you to navigate within the material or back to the very top of the material. The Nested Map drop-down list allows you to see the nested maps that are part of a material and to navigate to different parts of a material quickly.
- ✔ **Apply materials.** After a material is created, it can be assigned to a 3D object or an individual face by clicking the Apply Material to Objects button. The Remove Materials from Selected Objects button allows you to remove materials assigned to 3D objects and individual faces.
- ✔ **Purge materials.** Materials take up room in a drawing file, and if they are no longer needed you can purge a material that is no longer needed by clicking the Purge from Drawing button.

## Common material properties

Each type of material affects which properties change the visual characteristics of the material. Table 17-1 lists the properties and which type of material they're associated with.

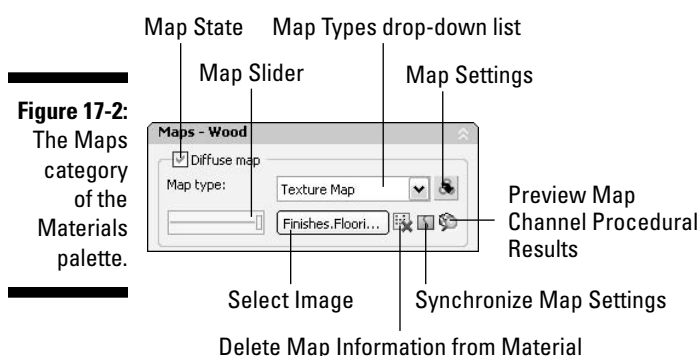
<i>Property Name</i>	<i>Material Type</i>	<i>Description</i>
Color/Diffuse Color	Realistic, Realistic Metal, Advanced, and Advanced Metal	Specifies the color of the material.
Ambient Color	Advanced and Advanced Metal	Determines the color of the material when lit by ambient light.
Specular Color	Advanced	Controls the highlight color for a material that has shininess.

(continued)

<b>Property Name</b>	<b>Material Type</b>	<b>Description</b>
Shininess	Realistic, Realistic Metal, Advanced, and Advanced Metal	Specifies the level of shininess a material has. The higher the value, the harder the material appears as it reflects more light.
Opacity	Realistic and Advanced	Controls how transparent a material is. The greater the opacity value, the less transparent the material is. A value of 0 designates that the material is completely transparent.
Refraction Index	Realistic and Advanced	Determines how light is refracted through an object that distorts the image picked up by the partially transparent material.
Reflection	Advanced Metal	Specifies how reflective a material is. The greater the value, the more reflective the material is. A value of 100 makes the material mirror-like, so it reflects all the objects around it.
Translucency	Realistic and Advanced	Controls how much light a material transmits and to what extent the light is scattered within an object.
Self-Illumination	Realistic, Realistic Metal, and Advanced	Determines whether and how much light is emitted from a material that is different from the user-defined lights in the drawing. When self-illumination is enabled, luminance isn't available.
Luminance	Realistic and Realistic Metal	Specifies the amount of light that a material reflects. When luminance is enabled, self-illumination isn't available.
Two-Sided Material	Realistic and Realistic Metal	Determines whether the material is applied to both sides of a face. By default, only the normal side of the face is assigned a material. The option is available only when Force Two-Sided is not turned on in the Advanced Render Settings palette. For more information on the Advanced Render Settings palette, see Chapter 19.

## Maps

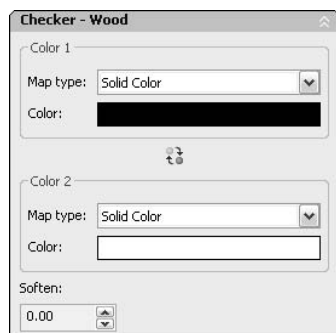
In addition to the properties that alter the way a material looks, you can define a material by using maps. Maps for a material are defined under the Maps category (see Figure 17-2) in the Materials palette. You can choose one of eight map types and up to four different maps for a material. You select a map type and then click the Map Settings button to access the settings for the selected map. The Map Slider allows you to determine the percentage of the map that is displayed with the Diffuse color.



You can select these eight map types:

- ✔ **Texture:** Allows you to use an image file as the texture for a material.
- ✔ **Checker:** Allows you to define a checker pattern with two different colors or map types.
- ✔ **Marble:** Allows you to define a swirl pattern that looks like the veins in a slate of marble. You can specify two different colors: One color is the vein color, and the other is the stone color.
- ✔ **Noise:** Allows you to create a pattern that has two different colors or map types and a level of turbulence that affects how the two colors or map types are mixed together.
- ✔ **Speckle:** Allows you to define a pattern of random dots by using two different colors and a size value for the speckles.
- ✔ **Tiles:** Allows you to create a repeating pattern of bricks and tiles by using two different colors or map types to represent the tile/brick color and the color of the grout. You can also define the size of the tiles/bricks and how they repeat.
- ✔ **Waves:** Allows you to create a random series of waves by using two different colors. You can specify properties like wave radius, number of waves, amplitude, and others to determine how the pattern is created. It looks similar to ripples on a pond.
- ✔ **Wood:** Allows you to define a wood-grain pattern that uses two different colors. One color is the wood color, and the other color is the grain.

Figure 17-3 shows the settings for a Checker procedural map.

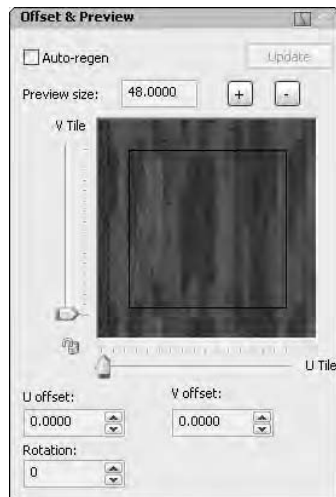


The four different map properties that you can choose are similar to the properties that you can set for a material:

- ✓ **Diffuse:** Use an image or procedural map to define the color or pattern that's displayed for the material.
- ✓ **Reflection:** Specify the reflectivity of a material and assign an image or procedural map that's reflected in the object that the material is assigned to. This property isn't available if you choose Realistic and Realistic Metal materials.
- ✓ **Opacity:** Define which parts of a material should be solid and which parts should be transparent. White areas are opaque, and darker areas are transparent.
- ✓ **Bump:** Create a three-dimensional pattern that appears textured or embossed based on a gray shaded image; darker areas appear embossed.

## Previewing materials and maps

You can see a preview of a material at the top of the Materials palette, but if you scroll down to the Material Offset & Preview category, you can see a much larger preview of the material. The Material Offset & Preview category allows you to see in real time what a material will look like without applying it to a 3D object and creating a rendering. You can also preview an individual map by clicking the Map Settings button and then scrolling down to the Offset & Preview category (see Figure 17-4). If you want to see the results of one of the maps only at the top level of a material, click the Preview Map Channel Procedural Results. The Map Preview dialog box opens, and you get a preview of the image or procedural map.



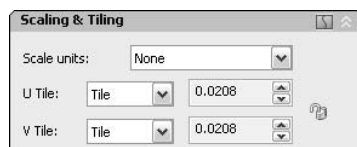
**Figure 17-4:**  
A preview of  
a map.

## Scaling, Tiling, and Offsetting Materials

When using maps, you often have to control how the material repeats or tiles. Tiling commonly wraps a 3D object or face with parts of an image over and over again. Scaling makes an image or procedural map fill or stretch across a 3D object or face. You might use scaling to place a logo or label on a part and tiling to create a repeating pattern like a wood pattern. Scaling can take place at the material or map levels, but scaling a material at the parent level allows you to specify a unit of measurement for

the scaling factor. Offsetting allows you to shift an image or procedural map to determine where the map starts and how it tiles. Figure 17-5 shows the Scaling & Tiling category that is displayed in the Materials Editor and is used to adjust the tiling, scaling, and offsetting of a material.

**Figure 17-5:** Scaling and tiling a map to control how the material repeats.



### Exercise 17-1: Creating Materials

For this exercise, open the drawing file `Exercisel7-1.dwg` from the Chapter 17 folder in the My Practice Drawings folder under My Documents. Then practice creating a few different materials and modifying the default material in the drawing.

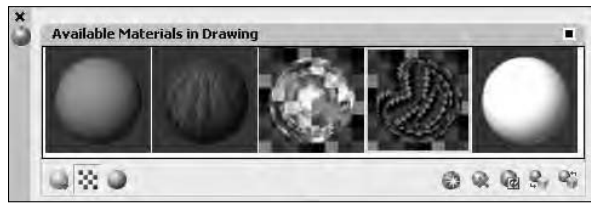


1. Choose Tools⇨Palettes⇨Materials to display the Materials palette.
2. On the Materials palette, click the Create New Material button.
3. In the Create New Material dialog box, enter **Wood** in the Name text box and add a description if you want to. Click OK.
4. With the new material created, slide the slider for the Shininess property so the text box reads 75.  
You can enter a value in the text box to the right of the slider and press Enter instead of using the slider.
5. Adjust the Refraction Index for the material to 1.200.
6. Under the Maps category, select Texture Map from the Map Type drop-down list under Diffuse Map if it's not already selected and click Select Image.
7. In the Select Image File dialog box, browse to the folder `C:\Documents and Settings\All Users\Application Data\Autodesk\AutoCAD 2008\R17.1\enu\Textures` and select the file `Finishes.Flooring.Wood.Hardwood.1.jpg`. Click Open.
8. Change the lighting model for the preview, click the Preview Swatch Lighting Model button, and select the bottom option.  
A back light is added to the preview.
9. Create another new material and give it the name Glass.
10. With the new material selected, select Glass – Clear from the Template drop-down list under the Material category.
11. Click the Color swatch just below the Template drop-down list to display the Select Color dialog box.
12. Click the True Color tab if it isn't active and then select RGB from the Color Model drop-down list. In the Red, Green, and Blue text boxes, enter the value 255. Click OK.

13. To better see the preview of the material, click the Checker Underlay toggle to add a checker pattern to the background of the material.
14. Create another new material and give it the name Mirror.
15. With the new material selected, select Realistic Metal from the Type drop-down list and then Mirror from the Template drop-down list under the Material category.
16. With the new material created, set the Shininess property to 100, set the color to the True Color 255,255,255, and also turn on the Checkered Underlay for the preview.
17. Create a new material and name it White. Set the color to the True Color 255,255,255 and set Self-Illumination to a value of 14.
18. In the Available Materials in Drawing category, select the Preview for the material named Global.
19. Set the Global material to use the color that the objects are assigned. Select the By Object check box to the right of the Color swatch.

The materials look like those shown in Figure 17-6.

**Figure 17-6:**  
Previews of  
materials  
created.



20. Save the drawing with the name `Completed17-1.dwg` to the Chapter 17 folder in the My Practice Drawings folder under My Documents and close the drawing.

For more information on creating materials and working with maps, see the topics Create Materials and Using Maps for Added Realism in the AutoCAD help system. To locate the topic, display the Online Help system and click the Contents tab. On the Contents tab, choose User's Guide⇨Create Realistic Images and Graphics⇨Materials and Textures.

## Accessing Libraries of Materials

It takes time to understand all the ins and outs of materials; that's why AutoCAD ships with a large number of materials. AutoCAD comes with two different sets of material libraries: one that's installed by default with AutoCAD and one you choose to install. You can find the material libraries from the Tool Palettes Window (choose Tools⇨Palettes⇨Tool Palettes) by right-clicking the title bar when the palette isn't docked or the double grip bar at the top when it is docked. Select either Materials or Materials Library from the contextual menu to display palettes that contain material tools.

The Materials tool palette group on the Tool Palettes Window contains some basic materials that you might use on your 3D models, whereas the Materials Library palette group (see Figure 17-7) contains a larger number of materials that you can attach to 3D objects in your 3D models. To attach a material, click one of the tools or click a tab to switch between the available tool palettes in the selected group to select a different

material. The material is added to the drawing and you're prompted to select the object you want to attach the material to. For more information on attaching materials, see the section "Attaching and Removing Materials," later in this chapter.

**Figure 17-7:**  
The Tool Palettes Window with the Materials Library tool palette group displayed.



You can create your own material tools on tool palettes if you want to reuse a material in other drawings. To reuse a material in another drawing, drag the preview swatch in the Materials palette to a tool palette on the Tool Palettes Window and drop the image when the black horizontal bar is displayed. For more information about customizing tool pallets, see the topic Tool Palettes in the AutoCAD help system. To locate the topic, display the Online Help system and click the Contents tab. On the Contents tab, choose User's Guide > The User Interface.

## Attaching and Removing Materials

Materials by themselves aren't any good until you attach them to an object in your 3D model. You can attach a material in three different ways: by layer, directly to an object, or to a surface on a 3D solid. Attaching materials to the layers is by far one of the easiest and quickest ways to attach materials to objects in your 3D models. However, using the layer approach to attach materials takes some up-front planning on your part. All objects on one layer receive the same material, so if you have different objects on one layer, you either have to create a new layer or directly attach a material to the objects that need a different material assigned to them.

To attach a material to a layer, use the MATERIALATTACH command or click the Attach by Layer button on the Materials control panel of the Dashboard palette. The Material Attachment Options dialog box (see Figure 17-8) opens. In the dialog box, drag a material from the left side to the layer you want to assign the material to on the right side. To remove the materials, click the Detach icon next to the layer and material pairing on the right side of the dialog box.

If you didn't give each object that requires a different material a layer, you can assign materials directly to individual objects. To assign a material to an object, display the Materials palette and then select the material you want to attach under the Available Materials in Drawing category. Then click the Apply Material to Object. Clicking Apply Material to Object starts the MATERIALASSIGN command and changes the crosshairs to a pick box with a paintbrush (see Figure 17-9). Select the object that you want to attach the material to.



**Figure 17-8:**  
The  
Material  
Attachment  
Options  
dialog box.

**Figure 17-9:**  
The  
paintbrush  
indicates a  
material is  
ready to be  
attached.



If your 3D model is created from 3D solids, you can assign a material to a specific face of a 3D solid. For example, you might have a 3D solid that has a metal material attached to it but you want to show a sticker material on a specific face. To assign the sticker material to a face, hold down the Ctrl key and select the face of the 3D solid you want to attach the material to. You can also use the Face option of the SOLIDEDIT command to assign a material by its name to a face of a 3D solid. (I cover the SOLIDEDIT command in Chapter 9.)



1. In the exercises in the chapter, you apply materials to 3D objects or faces. What steps do you take to create a 3D solid primitive of a cube-shaped box?

1. \_\_\_\_\_
2. \_\_\_\_\_
3. \_\_\_\_\_
4. \_\_\_\_\_



You can use the MATCHPROP command to assign a material to an object by using an existing object in the drawing that already has a material assigned to it. For more information on the MATCHPROP command, choose Modify→Match Properties and then press F1 when the command is started.



To remove materials from 3D objects or faces, click Remove Materials from Selected Objects on the Materials palette. When prompted to select objects, select the 3D objects that you want to remove materials from. The materials are removed from both 3D objects and individual faces on a 3D solid, so if you've assigned material to a face, it's also removed.



Materials can exist in a drawing even if they are not applied to a 3D object or face. In order to remove a material completely from a drawing, select a material in the Material Editor and click Purge from Drawing, or use the PURGE command.

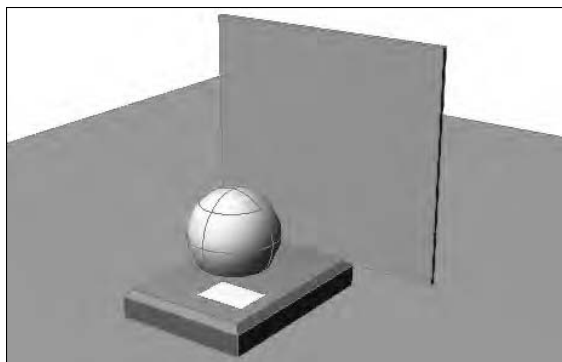


### Exercise 17-2: Attaching Materials to 3D Objects

For this exercise, open the drawing file `Exercise17-2.dwg` from the Chapter 17 folder in the My Practice Drawings folder under My Documents. Then practice attaching materials to objects and faces in the drawing.

1. Display the Dashboard and select Realistic from the Visual Styles drop-down list under the Visual Styles control panel.

The viewport looks similar to Figure 17-10.

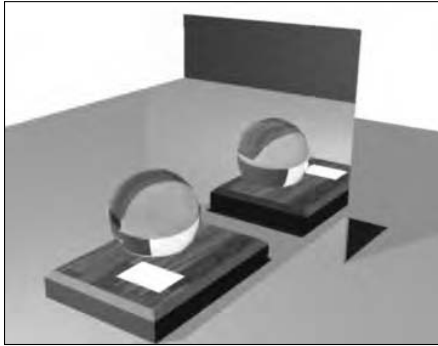


**Figure 17-10:**  
3D objects  
displayed  
with the  
Realistic  
visual style.

2. Display the Materials palette.
3. Select the Glass material preview and click Apply Material to Object.
4. At the `Select objects:` prompt, select the sphere in the drawing and press Enter.  
The sphere now appears transparent.
5. Select the Wood material preview and click Apply Material to Object.
6. At the `Select objects:` prompt, hold down the Ctrl key and select the face that's below the sphere.  
You might have to move the pick box near the 3D solid to get it to highlight first and then hold down the Ctrl key to select the face. AutoCAD can sometimes be picky about how you select the face. The top of the chamfered box now has a wood grain assigned to it.
7. Display the Dashboard if it's closed and click the Attach by Layer button on the Materials control panel.
8. In the Material Attachment Options dialog box, drag the Mirror material from the left side to the Mirror layer on the right. Do the same with the White material, except drop it on Label layer 0. Click OK.

9. Close the Materials and Dashboard palettes to see what the viewport looks like now. You really don't see all the materials to their true potential because AutoCAD can't produce some of the effects that materials make directly in the viewport.
10. Choose View⇨Render⇨Render.

The Render Window is displayed and shows what the objects look like after the materials have been attached (see Figure 17-11).



**Figure 17-11:**  
Objects rendered with materials attached.

11. Save the drawing with the name `Completed17-2.dwg` to the Chapter 17 folder in the My Practice Drawings folder under My Documents and close the drawing.

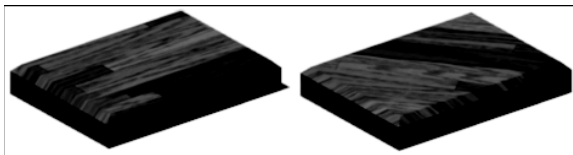
## Adjusting Materials

Materials can have repeating patterns such as ceramic tiles or grain in a wood material. The direction in which these materials run most likely is important in your rendering. When you attach a material, AutoCAD places it on the object based on the object's UVW coordinates — even if sometimes, the placement feels completely random. If you don't like how AutoCAD places the material, the MATERIALMAP command rotates and repositions a material on a 3D object or the face of a 3D solid. Start the MATERIALMAP command from the Mapping flyout on the Materials control panel or click one of the options from the Mapping submenu under View⇨Render.

When you've started the MATERIALMAP command, you're prompted to do one of three things:

- ✓ **Choose one of the four mapping gizmos.** The four gizmo options that you can choose are box, planar, spherical, and cylindrical shaped. The shape of the gizmo that you choose depends on the type of object you select and how you want the material to wrap. After you assign the gizmo, you can move or rotate it to change the mapping coordinates for the 3D object or face.
- ✓ **Copy a mapping.** You can copy the mapping coordinates from one 3D object or face to another one so you don't have to move or rotate the mapping over again on other objects.
- ✓ **Reset the material mapping for the 3D object or face.** You can also reset the mapping coordinates, which undoes any moving or rotating of the mapping gizmo. Figure 17-12 shows a hardwood flooring material rotated 45 degrees. The left image shows the material before it was rotated 45 degrees, and the image on the right shows the material after it was rotated.

**Figure 17-12:**  
A material rotated 45 degrees on an object.



Original material mapping      Material mapping rotated 45°

For more information on mapping materials, see the topic *Adjust Mapping on Objects and Faces* in the AutoCAD help system. To locate the topic, display the Online Help system and click the Contents tab. On the Contents tab, choose *User's Guide* → *Create Realistic Images and Graphics* → *Materials and Textures*.

### Exercise 17-3: Controlling the Direction of a Material

For this exercise, open the drawing file `Exercise17-3.dwg` from the Chapter 17 folder in the My Practice Drawings folder under My Documents. Then practice rotating a material.

1. Choose *View* → *Render* → *Mapping* → *Planar Mapping*.
2. At the *Select faces or objects:* prompt, hold down the Ctrl key, select the face with the wood material attached to it, and press Enter to end selecting objects.

You might have to move the pick box near the 3D solid to get it to highlight first and then hold down the Ctrl key to select the face. AutoCAD is sometimes picky about how you select the face.

3. At the *Accept the mapping or [Move/Rotate/reset/switch mapping mode]:* prompt, enter **R** and press Enter.

The same gizmo that's used for the 3DROTATE command is displayed.

4. Position the cursor over the blue ribbon on the gizmo and click when it turns yellow.

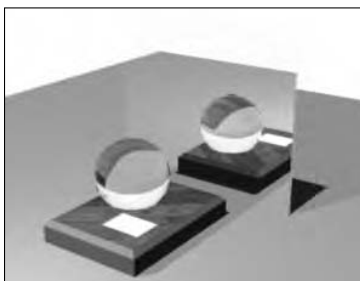
The gizmo is now locked to rotate along the Z axis.

5. As you move the crosshairs and the material starts rotating on the surface, enter **45** and press Enter twice to rotate the material 45 degrees.

6. Choose *View* → *Render* → *Render*.

The Render Window is displayed and shows what the rotated material looks like (see Figure 17-13). Compare it to Figure 17-11, which shows the material without any rotation.

**Figure 17-13:**  
A material on the face of the 3D solid rotated 45 degrees.



7. Save the drawing with the name `Completed17-3.dwg` to the Chapter 17 folder in the My Practice Drawings folder under My Documents and close the drawing.



2. Exercise 17-3 shows how to adjust the mapping coordinates for a face of a 3D object. What steps do you take to reset the mapping coordinates of a 3D object or face?

1. \_\_\_\_\_
2. \_\_\_\_\_
3. \_\_\_\_\_
4. \_\_\_\_\_

## Converting Old Materials

You save materials created in AutoCAD 2008 in a different format from the ones created in AutoCAD 2006 and prior. If you have a drawing that contains old materials and want to use them, you must use the `CONVERTOLDMATERIALS` command to change all the old materials in the drawing to the new type so you can access them from the Materials palette assigned to objects. If you don't convert the old materials, they won't be applied during rendering or shown with the Realistic visual style. If old materials are found and converted in the drawing, the number of materials that were converted is displayed in the AutoCAD Text Window. For example, the message `6 materials(s) converted.` indicates that six old materials were converted in the drawing.

Because the `CONVERTOLDMATERIALS` command can be a task in itself, you can have AutoCAD update old materials automatically for you. Set the system variable `3DCONVERSIONMODE` to a value of 1, which is the default value. If you set `3DCONVERSIONMODE` to 0, you disable the automatic conversion of old materials. Set the value of 2, which prompts you when old materials are available for converting. The `3DCONVERSIONMODE` system variable also affects the conversion of old lights in drawings created in AutoCAD 2006 and prior. To find out more about converting old lights, see Chapter 16.

## Answers to Practice Opportunities

1. **What steps do you take to create a 3D solid primitive of a cube-shaped box?**
  1. Choose `Draw` → `Modeling` → `Box`.
  2. Specify the first corner of the box.
  3. To specify the opposite corner of the box, use the `Cube` option to switch to cube creation mode.
  4. Specify the length and rotation angle of the cube.
2. **What steps do you take to reset the mapping coordinates of a 3D object or face?**
  1. If the Dashboard palette is not displayed, choose `Tools` → `Palettes` → `Dashboard`.
  2. Expand the Materials control panel on the Dashboard palette, position the cursor over the control panel's icon, and click. (The icon looks like a sphere with a paintbrush.)
  3. In the expanded section of the Materials control panel, click the `Reset Mapping Coordinates` button.
  4. When prompted to select objects, select the 3D objects that you want to reset the mapping coordinates for.

## Chapter 18

# Rendering and Animating

### *In This Chapter*

- ▶ Rendering a 3D model
- ▶ Using render presets
- ▶ Using advanced rendering settings
- ▶ Creating an animation

To generate a rendering of your 3D model, you don't need user-defined lighting or materials, but without those two elements, you won't generate realistic renderings of your 3D models. After adding user-defined lights and materials, you can create a rendering of the 3D model that you can easily save and e-mail or post on an extranet site. (An *extranet* is a Web site that is set up specifically for the client to access and view the progress of a project; the site is commonly protected by a password to make sure only authorized people have access to it.) AutoCAD tries to make rendering as easy as possible by hiding all the settings from you by using *render presets*. Render presets are a named collection of rendering settings that allow you to quickly create a rendering, but you can also work with the rendering settings directly to manipulate the rendered output.

Along with generating an image file, you can generate a basic walk-through or walk-around animation. The animation can help the client see all sides of a 3D model to get a better understanding of what it looks like and its scale. The animation capabilities that AutoCAD offers are somewhat limited, but they're better than no animation capabilities at all.

The exercises in this chapter show you how to create a rendering of your 3D model using different techniques.

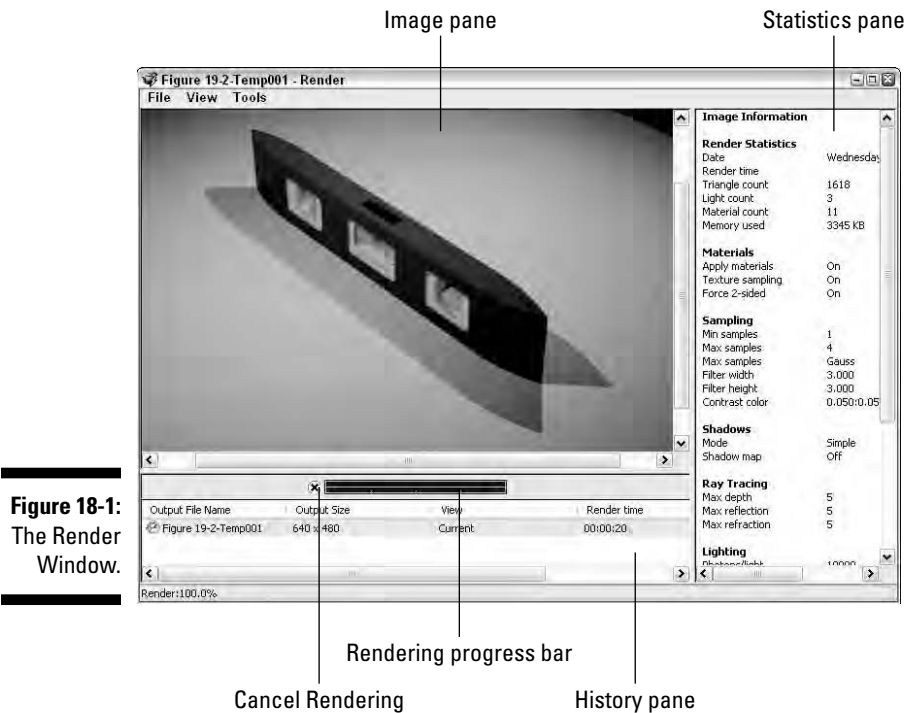
## *Rendering Your 3D Model*

The Render Window (see Figure 18-1) is the default location where AutoCAD renders the current view of your 3D model. The Render Window is displayed when you start the RENDER command, but you can also display it using the RENDERWIN command or by clicking Show Render Window when the Render control panel is expanded.

The Render Window gives you visual feedback about the current progress of the image:

- ✓ **See the render.** The Image pane is in the upper-left corner and displays the most recently rendered image.
- ✓ **Look at the history.** The History pane is located below the Image pane and allows you to view the other rendered images that you recently created. You can right-click and choose to render an image again, save a rendering to a file, remove the rendering from the history list, or simply delete the temporary file and remove the entry from the list.

The History pane gives you some information about how long the rendering took, the rendering preset used, the view rendered, the output size of the image rendered, and the name of the temporary file that AutoCAD created for the rendering. The temporary files that are created for each rendering are saved to the Temp folder of Windows.



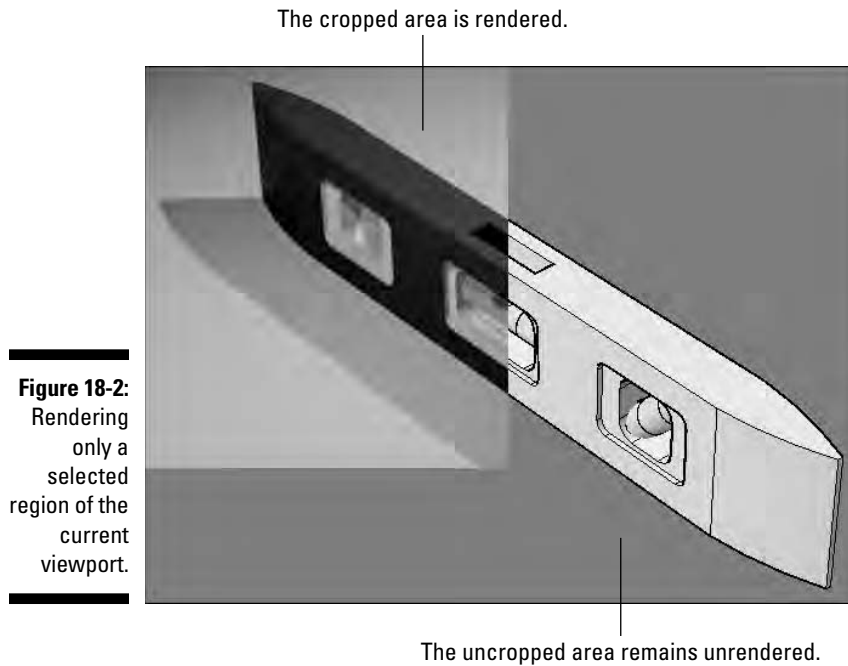
**Figure 18-1:**  
The Render Window.

- ✓ **Check the progress or cancel the render.** Between the Image and History panes are the Cancel Rendering button and the progress bar for the current rendering in progress. If you click Cancel Rendering when a rendering is in progress, the rendering is aborted. You can also see the current progress of a rendering from the Render control panel on the Dashboard, as well as abort the current rendering in progress.
- ✓ **Look into the statistics of a render.** On the right side of the Render Window is the Statistics pane, which gives you information about the settings used to create the rendering. These settings are stored in a render preset and can be changed with the Render Presets Manager dialog box or the Advanced Render Settings palette. I cover the Render Presets Manager dialog box later, in the section “Rendering with Presets,” and I cover the Advanced Render Settings palette later, in the section “Rendering with Advanced Settings.”

## *Cropping before a render*

Rendering can take a while when you’re working with complex 3D models that have a lot of materials attached to them and use a large number of user-defined lights. One of the things that you can do to help make rendering faster is to render only the part of the drawing window that you want to test your material or lighting changes in.

To render only a portion of the current viewport, use the `RENDERCROP` command. Click Render Cropped Region on the Render control panel or select Crop from the Procedure setting in the Advanced Render Settings palette before starting the `RENDER` command. Figure 18-2 shows a region of the current viewport rendered right in the drawing window, but the remaining part of the 3D model is left in the current visual style.



**Figure 18-2:**  
Rendering  
only a  
selected  
region of the  
current  
viewport.

## *Saving a rendering*

You can save renderings in a number of different file formats. Based on whether you created the rendering through the Render Window or the current viewport, you follow slightly different processes to save the rendered image to file.

When rendering to the Render Window, you right-click the temporary file in the History pane and choose Save. You give the rendering a name and specify a location.

If you want to save a rendering from the current viewport, you have to use a screen-capture tool like SnagIt by TechSmith ([www.techsmith.com](http://www.techsmith.com)) or the Print Screen feature that comes with Windows to get an image of the viewport. SnagIt offers a variety of screen-capture options and image-editing tools, but the Windows Print Screen feature is very basic. To create a print screen with Windows, press the Print Scrn (or Print Screen) key. An image is created and placed on the Windows Clipboard, which allows you to paste it into another application (such as Paint).



Pressing the Print Scrn (or Print Screen) key by itself captures the whole Windows desktop, whereas holding down the Alt key while pressing the Print Scrn (or Print Screen) key captures the active application.



### Exercise 18-1: Creating a Rendering and Saving It to an Image File

For this exercise, open the drawing file `Exercise18-1.dwg` from the Chapter 18 folder in the My Practice Drawings folder under My Documents. Then practice creating a rendering of a 3D model and then save the rendered image to file.



1. Display the Dashboard if it isn't already displayed.
2. On the Render control panel, click Render.
3. In the Render window, right-click the `Exercise18-1-Temp000` entry in the History pane. From the shortcut menu, choose Save.

The last three numbers at the end of the file might be different based on the number of temporary rendered images you have created.

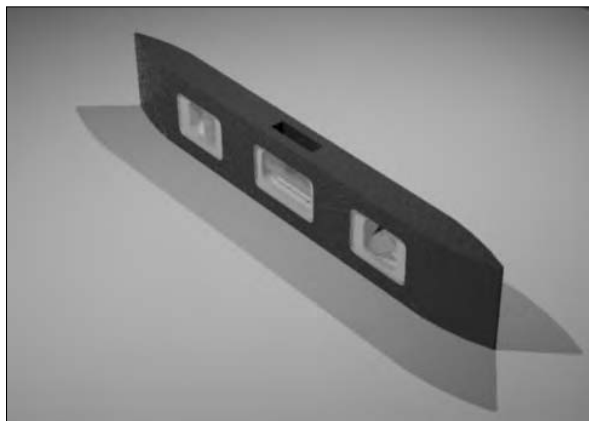
4. In the Render Output File dialog box, enter **Completed18-1** in the File Name text box and select PNG(\*.png) from the Files of Type drop-down list.
5. Browse to the Chapter 18 folder in the My Practice Drawings folder and click Save.
6. In the PNG Image Options dialog box, select 24 Bits (16.7 Million Colors) and deselect the Progressive check box. Click OK.

You return to the Render window. Notice that the icon for the temporary image file and its filename are changed to indicate that you saved the temporary file with a different name.

7. Close the Render window by clicking the X in the upper-right corner.
8. Browse to the Chapter 18 folder in the My Practice Drawings folder and double-click the `Completed18-1.png` file to view it the default application set up to view PNG files.

The rendered image looks similar to Figure 18-3.

**Figure 18-3:**  
An example  
of a rendered  
3D  
model with  
user-  
defined  
lighting and  
custom  
materials.



9. Close the drawing `Completed18-1.dwg` without saving any changes.

## Rendering with Presets

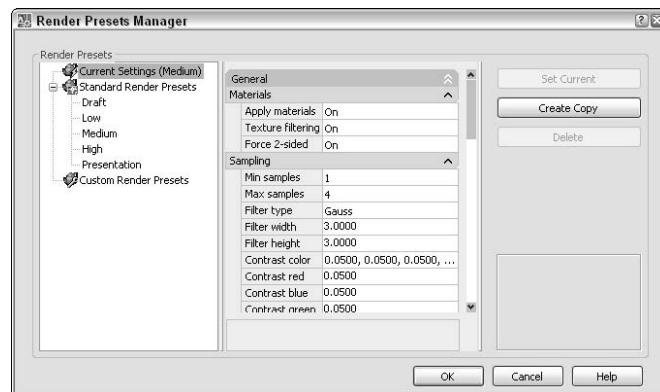
Prior to AutoCAD 2007, the rendering options were up front and center and often tempted you to click them in an attempt to find the perfect combination of settings to make a rendering look great. The set of scary rendering options is in the past since



Autodesk introduced render presets, which help simplify the rendering process. AutoCAD comes with five standard render presets that range from creating quick and low-quality renderings to high-quality renderings that are used for presentations.

Whenever you create a rendering, no matter whether the renderings destination is the Render window or the current viewport, you specify a render preset. The render preset tells AutoCAD which setting to use during rendering. To change the current render preset, you select one of the render presets from the Render Presets drop-down list on the Render control panel or from the Render Presets drop-down list on the Advanced Render Settings palette. Although the render presets that come with AutoCAD are great for most renderings, you can also create your own render presets by using the Render Presets Manager dialog box (see Figure 18-4). To display the Render Presets Manager dialog box, use the RENDERPRESETS command or select Manage Render Presets from the Render Presets drop-down list on the Render control panel.

**Figure 18-4:** You can create and manage render presets with the Render Presets Manager dialog box.



The Render Presets tree view on the left side of the Render Presets Manager dialog box allows you to select which render preset you want to view. You can work with the current rendering settings, view one of the standard render presets, or create a custom render preset. AutoCAD doesn't allow you to modify any of the standard render presets, but you can create your own custom render preset based on one of the standard render presets.

For more information on the rendering settings in the Render Presets Manager dialog box, see the topic RENDERPRESETS in the AutoCAD help system. To locate the topic, display the Online Help system and click the Contents tab. On the Contents tab, choose Command Reference → Commands → R Commands.



You can't export or import render presets, but if you want to add a specific render preset to a drawing, add it to a Drawing Template (DWT) file.

### Exercise 18-2: Creating and Using a Render Preset

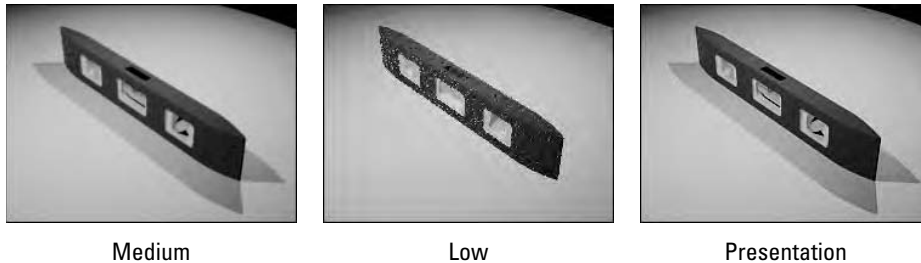
For this exercise, open the drawing file `Exercise18-2.dwg` from the Chapter 18 folder in the My Practice Drawings folder under My Documents. Then practice creating a custom render preset and using it to create a rendering.

1. Display the Dashboard if it isn't already displayed.
2. On the Render control panel, select Manage Render Presets from the Render Presets drop-down list.
3. In the Render Presets Manager dialog box, select Low under the Standard Render Presets in the Render Presets tree on the left side.

4. Click Create Copy.
5. In the Copy Render Preset dialog box, replace the text in the Name text box with **Quick Render** and click OK.
6. In the middle of the dialog box, click the vertical scroll bar to find the Shadows category. Click the lightbulb icon in the header of the Shadows category to disable the creation of shadows during rendering.
7. Use the scroll bar again and scroll all the way to the bottom. Click the Tile Order setting and then select Spiral from the drop-down list that's displayed.
8. Click Set Current to set the new render preset current and then click OK to exit the dialog box.
9. Create a rendering of the current view.

This time the rendering process starts in the center and works its way out. The quality of the image is much lower and doesn't have any shadows, which makes it quick to render but not very pretty to look at. The image that's created looks similar to the center image in Figure 18-5. The leftmost image is rendered with the Medium render preset, whereas the one on the right was rendered with the Presentation render preset.

**Figure 18-5:**  
The results  
of using the  
new Quick  
Render  
render  
preset.



10. Save the drawing with the name `Completed18-2.dwg` to the Chapter 18 folder in the My Practice Drawings folder under My Documents and close the drawing.



1. In Exercise 18-2, you created a new render preset and used it to create a rendering of the 3D model. To choose a different render preset, open the \_\_\_\_\_ control panel and select one from the \_\_\_\_\_ drop-down list.

## Rendering with Advanced Settings

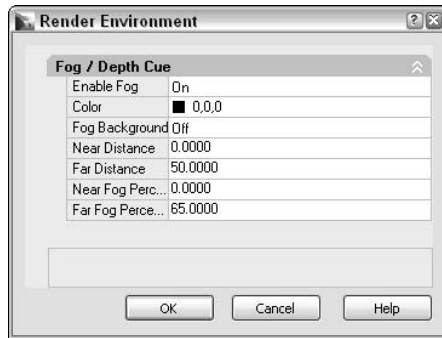
For the most part, the render presets AutoCAD provides for you are all you need. But if you want additional control over the rendering settings and how the final rendering looks, you should check out the advanced rendering features that allow you to improve the way your renderings look. These rendering features allow you to add fog and depth cueing to a rendering and to use advanced techniques such as Ray Tracing and Indirect Lighting.

### Render Environment

The Render Environment dialog box (see Figure 18-6) allows you to add fog and depth cueing to your renderings to give the viewer a sense of distance. You display the Render Environment dialog box with the `RENDERENVIRONMENT` command or by clicking the

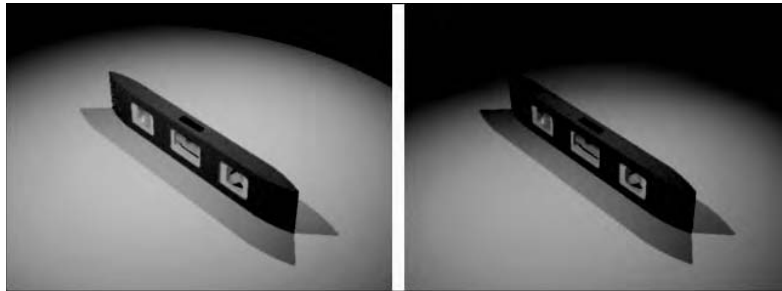
Render Environment button on the Render control panel on the Dashboard palette when it is expanded. To define fog or depth cueing, you specify a near and far distance with a percentage of opacity for the fog/depth cueing. The near and far distances are defined by a percentage from the Front Clipping Plane defined for the current view (or camera) towards the Back Clipping Plane. You can adjust the clipping planes by using either the View Manager dialog box when you're using a named view or the 3DCLIP command.

**Figure 18-6:** The Render Environment dialog box allows you to add fog or depth cueing to your renderings.



You can apply fog and depth cueing to the background in addition to the 3D objects in the drawing, which you normally need to do; otherwise, things look a little strange based on how your 3D model is being viewed. Figure 18-7 shows a rendering before and after depth cueing is added.

**Figure 18-7:** A model rendered with and without depth cueing applied.



Before depth cueing

After depth cueing

### Exercise 18-3: Adding Depth Cueing to a Rendering

For this exercise, open the drawing file `Exercise18-3.dwg` from the Chapter 18 folder in the My Practice Drawings folder under My Documents. Then practice changing the clipping planes of a named view and add depth cueing to a rendering of a 3D model.

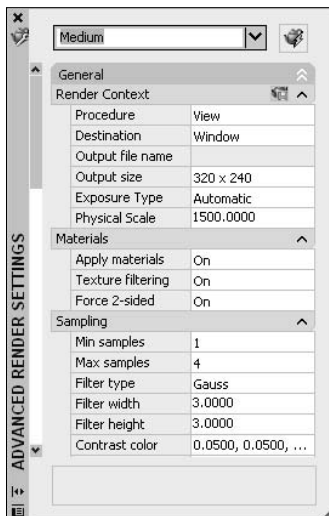
1. Choose View→Named Views to display the View Manager.
2. Select Render under the Model Views node in the Views tree.
3. In the middle of the dialog box, scroll to the bottom listing of available properties for the named view.
4. Under the Clipping category, enter a value of **10** for the Front Plane property and a value of **-10** for the Back Plane property. Select Front and Back On from the drop-down list for the Clipping property.

5. Click Set Current and then click OK.
6. Display the Dashboard palette if it isn't already displayed.
7. On the Render control panel, click the double down arrows to expand the Render control panel and then click Render Environment.
8. In the Render Environment dialog box, select the Enable Fog property and select On from the drop-down list.
9. Select the Color property and choose Select Color from the drop-down list.
10. In the Select Color dialog box, click the True Color tab and enter **0,0,0** in the Color text box at the bottom of the dialog box. Click OK.
11. Select the Far Distance property and enter **50** in the text box next to the property.
12. Select the Far Fog Percentage property and enter **65** in the text box next to the property.
13. Click OK to exit the Render Environment dialog box.
14. Render the drawing with the Medium render preset.  
The rendered results looks like the image on the right in Figure 18-7.
15. Save the drawing with the name `Completed18-1.dwg` to the Chapter 18 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Advanced Render Settings

The Render Presets Manager dialog box allows you to create custom render presets based on the values you want to use for the available rendering settings. However, you might at times want to change settings on the fly for unique situations or testing settings instead of creating a new render preset, or you might want to change the current context in which the rendering is created. The Advanced Render Settings palette (see Figure 18-8) allows you to change rendering settings and specify where the current rendering should be created or the file format that a rendering should be saved to. To display the Advanced Render Settings palette, use the `RPREF` command or click Advanced Render Settings on the Render control panel when it's expanded. For the most part, the Advanced Render Settings palette is a cross between some of the options found on the Render control panel and the Render Presets Manager dialog box.

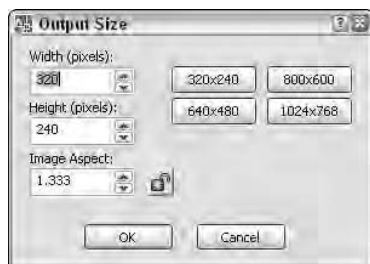
**Figure 18-8:**  
Exposing  
the  
advanced  
side of  
rendering  
with the  
Advanced  
Render  
Settings  
palette.



At the top of the Advanced Render Settings palette, you can choose a render preset or you can open the Render Presets Manager dialog box by selecting Manage Render Presets from the Render Presets drop-down list. The area that's different between the Render Presets Manager dialog box and the Advanced Render Settings palette is the Render Context category; otherwise, all the options are the same as those in the middle section of the Render Presets Manager dialog box. Here are the options to look at in the dialog box:

- ✓ **Procedure:** Select what you want to render. The procedures that you can choose are View (renders the current view in the drawing), Crop (starts the RENDERCROP command), and Selected (renders only the objects you select in the drawing window).
- ✓ **Destination:** Specify whether to use the Render Window or the current viewport (the Viewport option) to create the rendering. If you select Crop from the Procedures drop-down list, the only option you have for the Destination setting is Viewport; otherwise, both Window and Viewport are available.
- ✓ **Output File Name:** You see this setting only if you render to the Render window and click the Save File button located on the right side of the Render Context category header. Choose the default filename and file format that are used when a rendering is created. Each time a rendering is created, the most recent file is overwritten. Rendering to a file using this option is a nice safety net just in case AutoCAD crashes or you forget to save the last rendered image before you close AutoCAD.
- ✓ **Output Size:** Choose the size of the image that is created during rendering. Based on how the rendering will be used (on the Internet or in a printed brochure or flyer), you might need to adjust the size of the image that is created. AutoCAD defines four standard output sizes (320 x 240, 640 x 480, 800 x 600, and 1,024 x 768), but you can specify a custom image size by using the Output Size dialog box (see Figure 18-9), which you display by selecting Specify Output Size from the Output Size drop-down list. You can also specify the output filename and size for a rendered image by using the set of controls at the bottom of the Render control panel when it's expanded.

**Figure 18-9:**  
Specifying a  
custom  
image size.



The final options deal with the Render Exposure feature, which I cover in Chapter 16. When you select Automatic from the Exposure Type setting, AutoCAD determines the render exposure to use, but the settings that are set using the Render Exposure dialog box are used when Exposure Type is set to Logarithmic. The Physical Scale setting controls the scale factor applied to the values specified in the Adjust Render Exposure dialog box or the sliders on the Lights control panel when it's expanded.

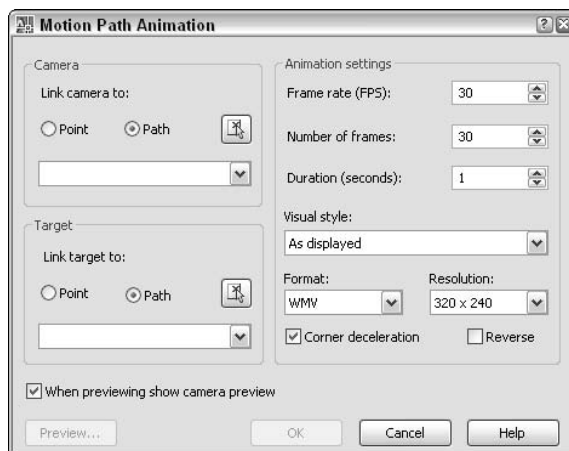
For more information on the other advanced settings in the Advanced Render Settings palette, see the topic Advanced Render Settings Palette in the AutoCAD help system. To display the topic, display the Advanced Render Settings palette and press F1 when the palette has focus.

## Setting Your Model in Motion

Renderings are a great way of showing how a design concept looks if it was built or manufactured, but they're still flat images of a 3D model. Although it isn't possible to pull your 3D model out of the screen into the real world (it would be great if you could, though) and allow your client to rotate it to see multiple points of view, you can opt for the next best thing. Animations in AutoCAD give you the ability to take your client on a visual tour of a 3D model or an assembly of parts. Although you can't really animate parts in motion, creating an animation from a camera rotating around a 3D model can help add something to a presentation and close the bid on a project. If you want to animate individual parts for animations that show an assembly coming together or explode a part to show how it goes together, you need a program like Autodesk Viz or 3ds Max. To find out more about Autodesk Viz or 3ds Max, visit Autodesk's Web site at [www.autodesk.com](http://www.autodesk.com).

To create an animation in AutoCAD, use the ANIPATH command or choose View⇨ Motion Path Animations to open the Motion Path Animation dialog box (see Figure 18-10). The Motion Path Animation dialog box allows you to control the motion of a camera around or through a 3D model. You designate whether the camera for the animation should follow a path or lock to a fixed point. The path can be a line, a circle, an arc, a 2D or 3D polyline, a spline, an ellipse, or an elliptical arc. You also specify how the target for the animation should be defined by using either a path or point. You can use only a point for the camera or target, but not both. Exercise 18-4 shows how to create an animation with the Motion Path Animation dialog box.

**Figure 18-10:**  
The Motion Path Animation dialog box allows you to create animation.

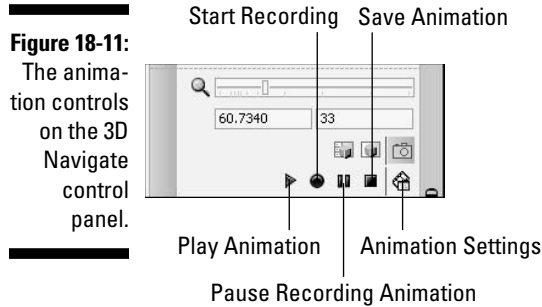


You then specify the settings for the animation that is generated. On the right side of the Motion Path Animation dialog box, you specify the number of frames per second, the length of the animation, visual style to be used, the size and format of the animation file to be created, whether the animation is slowed down when the camera is rounding a corner to make sure the animation appears smooth, and whether you want to reverse the direction that the animation is created in.

In addition to the Motion Path Animation dialog box, you can create an animation by using the controls on the 3D Navigate control panel when it's expanded (see Figure 18-11). Here's how you use the controls on the 3D Navigate control panel:

- ✓ **Record:** You start by using 3DWALK, 3DFLY, or another 3D navigation command and then click Start Recording Animation.
- ✓ **Save:** Click Save Animation.

✔ **Control the output:** Click Animation Settings to open the Animation Settings dialog box. It contains some of the same options that are in the Animation Settings section of the Motion Path Settings dialog box.



**Figure 18-11:** The animation controls on the 3D Navigate control panel.



2. Exercise 18-4 shows how to create an animation through the Motion Path Animation dialog box. What steps do you take to create an animation through the 3D Navigate control panel on the Dashboard palette?

1. \_\_\_\_\_
2. \_\_\_\_\_
3. \_\_\_\_\_
4. \_\_\_\_\_
5. \_\_\_\_\_
6. \_\_\_\_\_
7. \_\_\_\_\_

### Exercise 18-4: Following the Path to Create an Animation



For this exercise, open the drawing file `Exercise18-4.dwg` from the Chapter 18 folder in the My Practice Drawings folder under My Documents. Then practice creating an animation where the camera follows a path and the target is defined by a fixed point.

1. Choose View → Motion Path Animations.
2. In the Motion Path Animation dialog box, select Path under the Camera section and then click Select Path.
3. In the drawing window, select the spline for the path.
4. In the Path Name dialog box, replace the name in the Name text box with **Spline** and click OK.
5. In the Motion Path Animation dialog box, select Point under the Target section and then click Pick Point.
6. At the Pick point: prompt, enter **8.2,-18.5,1** and press Enter.
7. In the Point Name dialog box, click OK.

8. In the Motion Path Animation dialog box, under the Animation Settings section, replace the value in the Number of Frames text box with **600**. Also, select Conceptual from the Visual Styles drop-down list.

The other options under the Animations Settings section should be as follows:

- Frame Rate (FPS) = 30
- Format = WMV
- Resolution = 320 x 240
- Corner Deceleration = Selected
- Reverse = Deselected

9. Click OK.
10. In the Save As dialog box, save the animation with the name `Completed18-4` to the Chapter 18 folder in the My Practice Drawings folder under My Documents. Click Save.  
  
The Creating Video and Animation Preview dialog boxes are displayed. They show you the current progress of the animation that's being created.
11. After the animation is created, browse to the Chapter 18 folder in the My Practice Drawings folder and double-click the `Completed18-4.wmv` file to view it the default application set up to view MOV files.
12. Try creating other animations by using your own paths and other visual styles.
13. Close the drawing `Completed18-4.wmv` without saving any changes.

## *Answers to Practice Opportunities*

1. To choose a different render preset, open the Render control panel and select one from the Select Render Presets drop-down list.
2. What steps do you take to create an animation through the 3D Navigate control panel on the Dashboard palette?
  1. On the Dashboard palette, click the panel icon (Compass icon) located on the left side of the 3D Navigate control panel to expand it.
  2. Start a 3D navigation command, such as 3DZOOM, 3DWALK, 3DFLY, or 3DPAN.
  3. On the 3D Navigate control panel, click the Start Recording Animation button (red circle icon) to start recording.
  4. Use the 3D navigation command and move around the drawing.
  5. On the 3D Navigate control panel, click the Save Animation button (black square icon) to stop creating the animation and to save it to a file.
  6. In the Save As dialog box, enter a name and location for the animation file and then click Save.
  7. Exit the 3D navigation command.



## Chapter 19

# Additional Exercises: Visualizing the Design

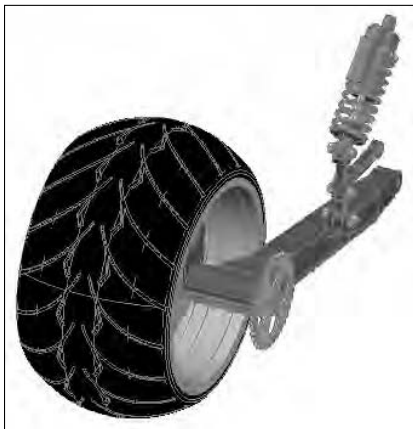
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### *In This Chapter*

- ▶ Adding more light
  - ▶ Creating and attaching more materials
  - ▶ Rendering another model
- 

**I**n the previous chapters of Part V, you find out how to create and modify lights and materials, along with how to create rendered images and animations. The exercises in this chapter build on the 3D modeling and navigation exercises from Chapters 7, 11, and 15. During the exercises in this chapter, you add lights and materials to the 3D model. Then you add some depth cueing and create a presentation-quality rendering that you save to a file.

The drawing that you work with uses the drawings that you created in Chapters 11 and 15. A planar surface has been created below the rear assembly of the custom motorcycle. The planar surface will be used to receive shadows and light. Some materials have already been defined for you and attached to some of the layers that the 3D objects are on. Figure 19-1 shows the rear assembly of the custom motorcycle that you will work with.



**Figure 19-1:**  
The rear  
assembly of  
a custom  
motorcycle.

Model courtesy of Mark Hechel of MasterGraphics, Inc.

## *Lighting the Way*

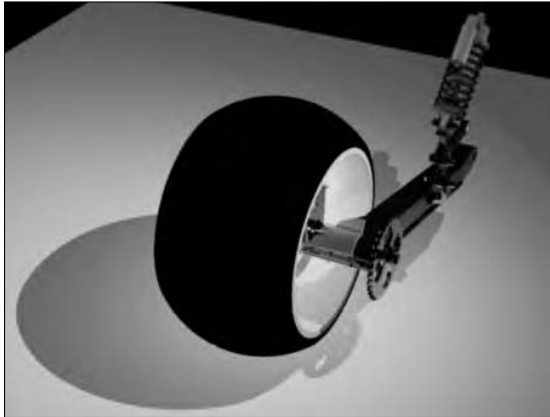
In this exercise, you add lights to the 3D model to give it a splash of realism. You add a point light, spotlight, and distant light to the 3D model. The spotlight is positioned at an angle almost directly above the model; the point light adds some lighting from inside of the rim of the model. The distant light brings some overall fill light to the entire rendering.



### Exercise 19-1: Adding User-Defined Lights

For this exercise, open the drawing file `Exercise19-1.dwg` from the Chapter 19 folder in the My Practice Drawings folder under My Documents. Then practice creating a point light, a spotlight, and a distant light.

1. Set LIGHTINGUNITS to a value of 2.
2. Display the Dashboard if it isn't already displayed and expand the Lights control panel.
3. On the Lights control panel, click Create a Point Light.
4. If the Viewport Lighting Mode dialog box is displayed, click Yes to switch lighting modes.
5. When prompted for a source location of the point light, enter **1.68,0.34,0.53** and press Enter twice.
6. On the Lights control panel, click Create a Spotlight.
7. When prompted for a source location of the spotlight, enter **38.5,-5.3,54** and press Enter.
8. For the target location of the spotlight, enter **-18.5,-5.3,-14** and press Enter twice.
9. Choose View⇨Render⇨Light⇨New Distant Light. If the Photometric Distant Light message displays, click OK.
10. When prompted for the FROM direction of the distant light, enter **77,-7,54** and press Enter.
11. For the TO direction of the distant light, enter **3.4,-5,0** and press Enter twice.
12. Display the Lights in Model palette.
13. On the Lights in Model palette, select Pointlight1 and right-click. From the shortcut menu, choose Properties.
14. On the Properties palette, set the properties for the point light to the following:
  - Name = Accent - Rim
  - Shadows = Off
  - Intensity Factor = 0.05
15. Select Spotlight2 from the Lights in Model palette and use the Properties palette to set the properties for the spotlight to the following:
  - Name = Overhead
  - Hotspot Angle = 35
  - Falloff Angle = 80
  - Intensity Factor = 3.0
16. Select Distantlight3 from the Lights in Model palette and use the Properties palette to set the properties for the distant light to the following:
  - Name = Area Fill
  - Shadows = Off
  - Intensity Factor = 0.1
17. Create a rendering of the 3D model using the Medium render preset to get a better look at the lighting that you just created.  
The rendered image that is created looks similar to Figure 19-2.
18. Save the drawing with the name `Completed19-1.dwg` to the Chapter 19 folder in the My Practice Drawings folder under My Documents and close the drawing.



**Figure 19-2:**  
A rendering  
created  
with user-  
defined  
lighting.

Model courtesy of Mark Hechel of MasterGraphics, Inc.

## Creating and Attaching Materials

In this series of exercises, you create a couple of new materials to use with the assembly and then attach them to 3D objects. First you create a rubber compound and chrome material. Then you attach the rubber compound material directly to the tire of the assembly and attach the chrome material to a few different layers. After that, you attach a material that has already been created for you to the face of one of the 3D objects.

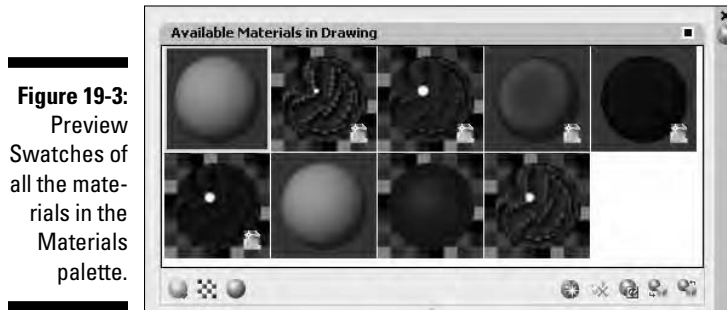


### Exercise 19-2: Creating Rubber Compound and Chrome Materials

For this exercise, open the drawing file `Exercise19-2.dwg` from the Chapter 19 folder in the My Practice Drawings folder under My Documents. Then practice creating two different materials that attach to objects in the assembly.

1. Display the Materials palette if it isn't already displayed.
2. Click Create New Material. Use Rubber Compound for the name of the new material and enter a description if you want to.
3. With the new Rubber Compound material selected, change the following properties for the material:
  - Type = Advanced
  - Ambient Color = 105,105,105
  - Diffuse Color = 0,0,0
  - Specular Color = 26,26,26
  - Shininess = 10
  - Self-Illumination = 6
4. Create another new material and give it the name Chrome. If you want, enter a description as well.
5. With the new Chrome material selected, change the following properties for the material:
  - Type = Realistic Metal
  - Template = Metal
  - Color = 173,173,173
6. Turn on the checkered background for both materials that you just created.

- The Preview Swatches in the Materials palette look similar to the ones in Figure 19-3.



**Figure 19-3:**  
Preview Swatches of all the materials in the Materials palette.

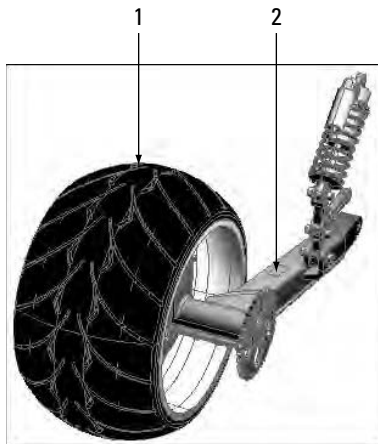
- Save the drawing with the name `Completed19-2.dwg` to the Chapter 19 folder in the My Practice Drawings folder under My Documents and close the drawing.

### Exercise 19-3: Attaching Materials



For this exercise, open the drawing file `Exercise19-3.dwg` from the Chapter 19 folder in the My Practice Drawings folder under My Documents. Then practice attaching materials to 3D objects in the drawing by selecting an object and a face on a 3D solid and attaching a material to several layers.

- Display the Materials palette if it isn't already displayed.
- Select the Rubber Compound material that you created in Exercise 19-2 and click Apply Material to Object.
- When prompted to select objects to attach the material to, select the tire indicated by 1 in Figure 19-4. Press Enter to complete the command.



**Figure 19-4:**  
Attaching materials to the rear assembly.

*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

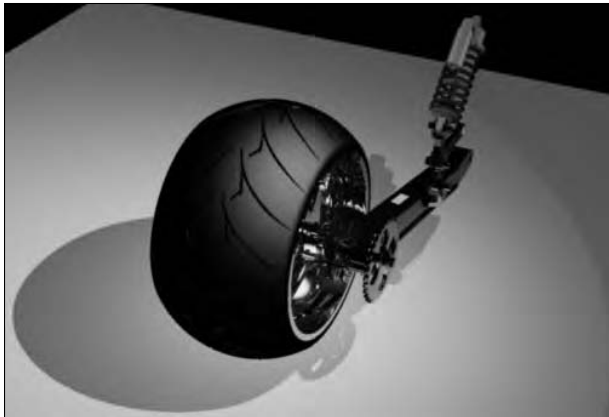
- Select the Label material in the Available Materials in Drawing category (near the Rubber Compound material). The Label material is displayed as a gray color.
- Click Apply Material to Object and when prompted to select objects to attach the material to, hold down the Ctrl key and select the rectangular face indicated by 2 in Figure 19-4.

The rectangular face should turn gray. If not, undo the material assignment and try again.

6. Display the Dashboard if it isn't already displayed and click Attach By Layer under the Materials control panel.
7. From the left side of the Material Attachment Options dialog box, drag the Chrome material onto the layers Rim, Rim\_hub, Rim\_fasteners, and Brake\_hub. Click OK to close the dialog box.
8. Create a rendering of the 3D model using the Medium render preset to get a better look at the lighting and the materials that you just created.

The rendered image looks similar to Figure 19-5.

**Figure 19-5:**  
A rendering  
created with  
user-  
defined  
lighting and  
attached  
materials.



Model courtesy of Mark Hechel of MasterGraphics, Inc.

9. Save the drawing with the name `Completed19-3.dwg` to the Chapter 19 folder in the My Practice Drawings folder under My Documents and close the drawing.

## Rendering the Model

In this exercise, you add depth cueing to the 3D model and create a presentation-quality rendering. To add depth cueing, you use the Render Environment dialog box and generate the presentation-quality rendering with the Presentation render preset.

### Exercise 19-4: Adding Depth to the Model and Creating a Presentation-Quality Rendering

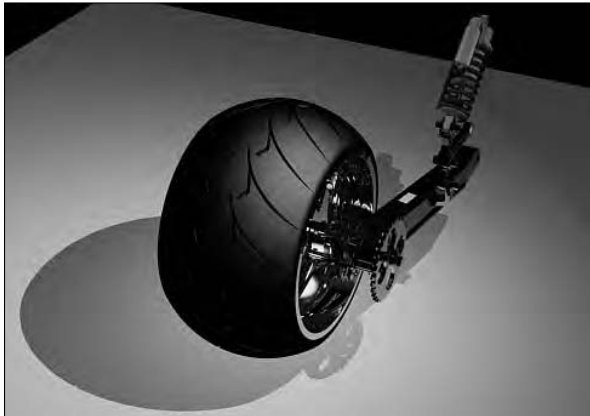


For this exercise, open the drawing file `Exercise19-4.dwg` from the Chapter 19 folder in the My Practice Drawings folder under My Documents. Then practice adding depth cueing to the model and create a presentation-quality rendering.

1. Display the View Manager dialog box (View⇨Named View) and select the Render named view under the Model Views node in the Views tree. Scroll down to the bottom of the Properties section (in the middle) of the dialog box and set the following options under the Clipping category:
  - Front Plane = 6.4
  - Back Plane = -109.0
  - Clipping = Front and Back On
2. Select the Render view, click Set Current, and then click OK.

3. Display the Dashboard if it isn't already displayed and expand the Render control panel.
4. On the Render control panel, click Render Environment.
5. In the Render Environment dialog box, set the options to the following values:
  - Enable Fog = On
  - Color = 0,0,0
  - Far Distance = 55.0
  - Far Fog Percentage = 35
6. On the Render control panel, select Presentation from the Render Presets drop-down list and click Render.
7. In the Render Window, right-click the most recently created image in the History pane and select Save.
8. Save the image to the Chapter 19 folder in the My Practice Drawings folder under My Documents and give the name `Completed19-4.png`. Select the PNG file format and click Save. Select the desired images in the Image Options dialog box or keep the default values, and click OK.
9. Browse to the location that the image was saved to and double-click the `Completed19-4.png` file to view it in the default image viewer installed on your computer.

The rendered image looks similar to Figure 19-6. If you need to, adjust the Intensity Factors of your lights to make the rendering of the 3D model appear darker or brighter based on the materials. A rendering might look different from computer to computer and from screen to print.



**Figure 19-6:**  
Final rendered image of the assembly.

*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

10. Save the drawing with the name `Completed19-4.dwg` to the Chapter 19 folder in the My Practice Drawings folder under My Documents and close the drawing.

---

### Exercise 19-5: Creating an Animation

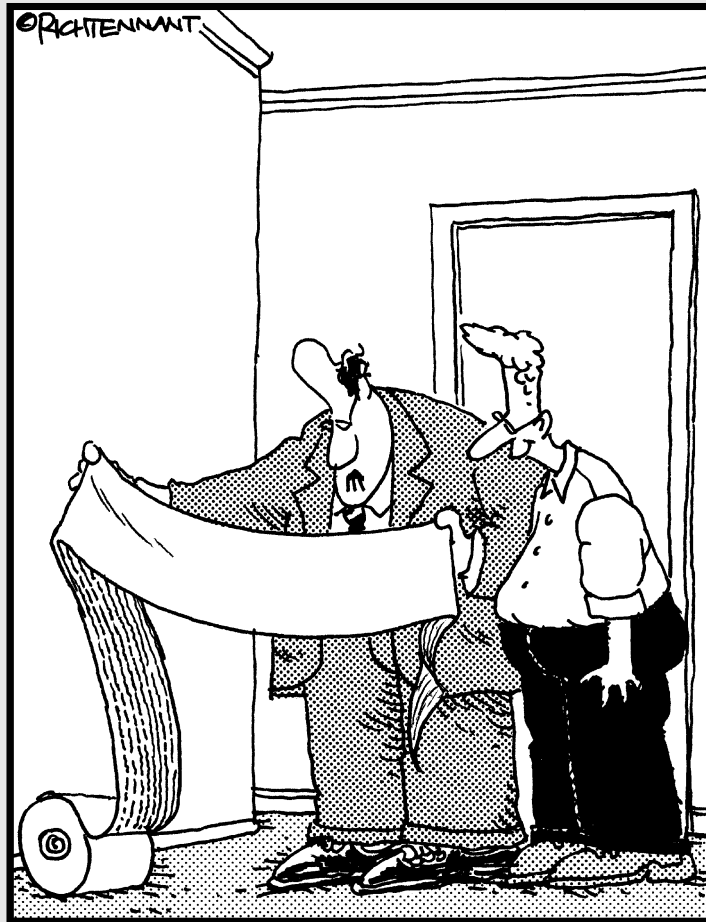
Try creating an animation by using the drawing file `Completed19-3.dwg` (or if you didn't complete Exercise 19-3, you can use the drawing file `Exercise19-3.dwg`) from the Chapter 19 folder in the My Practice Drawings folder under My Documents. Use either the Motion Path Animation dialog box or the controls on the 3D Navigate control panel to create the animation.

# Part VI

# Model Interchange

The 5<sup>th</sup> Wave

By Rich Tennant



"That's the first time I've seen someone use AutoCAD to construct a sentence."

### *In this part . . .*

**C**reating 3D models is great for showing design concepts as well as detecting any possible issues with a design before a prototype is ever created. However, because of the way collaboration on designs works these days, you must be able to share your designs electronically and in hardcopy formats with others. AutoCAD allows you to exchange 3D models in many different formats that non-CAD and CAD users can accept to keep using the information that you generated downstream. You can plot 3D models in many different ways, just like you can 2D drawings. When you plot a drawing, typically it's to paper, but you can also plot to electronic file formats.



## Chapter 20

# Plotting to a Hardcopy or an Electronic File

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### *In This Chapter*

- ▶ Plotting styles and plotters
  - ▶ Laying out your models
  - ▶ Creating a hardcopy of a model through plotting and publishing
  - ▶ Publishing your models to the Web
- 

**A**lthough you do live in the digital age, it's certainly not the paper-free utopia that many thought it would be with the birth of the Internet and e-mail. Paper still rules the roost when it comes to business transactions for the most part. Now with design software, the world has embraced sending files electronically, and people are working electronically a little more and more each year, but paper still plays an intricate part in a final design. When you want to share your 3D model with someone, you have two choices: keep them electronic (which the trees would be happy about) or create a *hardcopy* (print a drawing on a sheet of paper).

No matter whether you decide to keep your design in an electronic format or create a hardcopy, you are either plotting or publishing your design in some form to share it with others. Plotting or publishing your models gives you better protection of your files if you need to keep the original drawing files out of the hands of individuals outside your company. You can also exchange Drawing (DWG) files with AutoCAD users or other applications. I cover exchanging DWG files with users of other software programs in Chapter 21.

## *Plotters and Plot Styles*

Many things affect how you create an electronic representation of a 3D model or a hardcopy. The two things that affect the way a drawing file is output the most are plot styles and plotters. If a viewport has either the 3D Wireframe or 2D Wireframe visual style applied, the edges of 3D objects are affected by the plot style associated with the layout or the one selected during plotting. (Plot styles don't impact the way 3D models are plotted to an electronic file or hardcopy when they're displayed in a viewport with one of the standard 3D visual styles — 3D Hidden, Conceptual, and Realistic.) *Plotters* are the devices that produce output with the Plot or Publish dialog boxes to create an electronic file or a hardcopy.

### *Plotters*

Plotters control the output your 3D model is plotted to. A plotter can represent a system printer that's configured through Windows or a non-system printer. The settings for a non-system printer are stored in a plot configuration file. The plot configuration file represents

either a physical or virtual device that instructs AutoCAD about how to handle plotted output. Plot configuration settings are stored in files with the extension PC3 and are stored in the Plotters folder. (To locate the Plotters folder on your computer, open the Options dialog box and click the Files tab. Expand the Printer Support File Path node and then expand the Printer Configuration Search Path node. You can also choose File→Plotter Manager, which opens the folder that the plotter configuration files are stored in.)

A *physical device* is some sort of output machine like a laser jet printer or large format plotter that's connected directly to your computer or available from a network; a *virtual device* is a piece of software that runs on your computer and converts plotted output from AutoCAD to an electronic file. You create DWF, PDF, and PLT files by using virtual devices. Each PC3 file that you configure in AutoCAD works with one and only one plotter or file format at a time. AutoCAD ships five PC3 files out of the box, and for the most part, they're all virtual devices with the exception of the Default Windows System Printer .pc3 file, which is a special PC3 file that acts like a chameleon: It can represent either a physical or virtual device because it simply sends plot information to whatever the default Windows printer is that's currently configured on your computer.

To create a new PC3 file, you use the Add-A-Plotter Wizard shortcut in the Plotters folder. (You can access the Plotters folder from AutoCAD by choosing File→Plotter Manager.) When the Add Plotter Wizard is started, you can configure a new plotter that is connected to your computer or located on the network. If you configure a plotter on your computer, you have the ability to create PC3 files that output to DWF, PDF, DXB, and some common raster image file formats. Being able to plot directly to an electronic file makes your files secure by not allowing someone the ability to alter the original file, but you can also share them with non-CAD users. I cover other ways of creating raster image in Chapter 21.



### Exercise 20-1: Creating a Raster Image Plotter and Plotting to an Image File

In this exercise, you create a PC3 file that will allow you to plot directly to a Windows Bitmap (BMP) file.

1. Choose File→Plotter Manager.
2. In the Plotters folder, double click the shortcut Add-A-Plotter Wizard.
3. In the Add Plotter Wizard, click Next at the Introduction Page.
4. On the Begin page, select My Computer and click Next.
5. On the Plotter Model page, select Raster File Formats from the Manufactures list and then MS-Windows BMP (Uncompressed DIB) from the Models list. Click Next.
6. On the Import Pcp or Pc2 page, click Next.

PCP files are similar to the PC3 files except they were used with AutoCAD R12 and R13, whereas PC2 files were used with AutoCAD R14.

7. On the Ports page, make sure Plot to File is selected and then click Next.
8. On the Plotter Name page, replace the name in the Plotter Name text box with **BMP File** and then click Next.
9. On the Finish page, click Finish.

If you want to, you can click Edit Plotter Configuration to access the custom properties that are available for the device.

10. Close the window for the Plotters folder by clicking the X in the upper-right corner of the window.



1. Exercise 20-1 shows how to add a new plotter that allows you to plot to a BMP file. To configure a plotter to create a DBX file, select \_\_\_\_\_ from the Manufacturers list box.

## Plot styles

Plot styles aren't really useful when you're plotting or publishing 3D models because often you plot a viewport that's displayed with a 3D visual style (3D Hidden, Conceptual, or Realistic) or one that has been rendered. However, if you have a combination of viewports that are set up to display 2D and 3D visual styles, you control the way 2D objects appear with a plot style. In the case of 2D objects, you can use plot styles to change color, linetype, and lineweight for the linework in your drawing. When you create a new drawing, you designate the type of plot style you want to use when plotting. AutoCAD supports two different types of plot styles:

- ✓ **Color-dependent:** These plot styles allow you to assign a different color, linetype, and lineweight based on the color an object is assigned directly or through the layer it is on. Color-dependent plot styles are stored in CTB files.

Color-dependent plot style is the more popular of the two plot styles, most likely because it's the older of the two. The other reason why color-dependent plot styles are more popular is that dimensions and tables don't fully support named plot styles.

- ✓ **Named:** These plot styles use names to organize settings like color, linetype, and lineweight that are used when plotting objects. Named plot styles aren't automatically assigned to layers or objects like color-dependent plot styles are, so you must select the named plot style that you want a layer to use when a drawing is plotted or assign a named plot style directly to objects in your drawing. Named plot styles are stored in files with the extension STB.

Plot styles are stored in the Plot Styles folder. To locate the Plot Styles folder on your computer, open the Options dialog box and click the Files tab. Expand the Printer Support File Path node and then expand the Plot Style Table Search Path node.



If you want to convert a drawing that uses color-dependent plot styles to one that uses named plot styles, you use the CONVERTPSTYLES command (which is available only at the command prompt).

For more information about how to create and use plot styles, see the topic Use Plot Styles to Control Plotted Objects in AutoCAD's Online Help system. To locate the topic, click the Contents tab and then choose User's Guide⇨Plot and Publish Drawings⇨Plot Drawings⇨Control How Objects Are Plotted⇨Use Plot Styles to Control Plotted Objects.

## Laying Out Models

AutoCAD drawings contain two different locations that you can plot from: Model Space and Paper Space. Model Space is defined by the Model tab at the bottom of the drawing window or the Model button on the status bar when tabs aren't displayed. You do

all of your modeling on the Model tab; plotting is usually handled from the layout tabs. All the tabs in the drawing other than Model represent Paper Space that organizes how your drawing will be plotted (which is why it's called Paper Space). To lay out your model on the layout tabs, you use floating viewports that allow you to see portions of the model that are contained on the Model tab.



To display the Model and layout tabs on the status bar instead of at the bottom of the drawing window, right-click a tab and select Hide Layout and Model Tabs. To redisplay the Model and layout tabs, right-click the Model and layout buttons on the status bar and select Display Layout and Model Tabs.

## Layouts

You create *layouts* to organize different views of the model in Model space. Layouts are named objects that contain various settings that control how the objects on the layout and the views of the model are displayed in the viewports when plotted. Typically, a layout contains a title block, some text and dimensions, and objects called floating viewports. (I cover floating viewports later in this chapter.) The title block contains drawing and project-related information that helps organize books of drawings into sheet sets, which helps manufacture or assemble the manufactured parts.

In addition to holding graphical objects, layouts organize nongraphical information that tells AutoCAD how the layout tab should be plotted through the use of a page setup. A page setup contains all the settings required to plot or publish a layout such as the plotter, paper size, plot scale, plot style, and many other settings. The Page Setup Manager dialog box is used to create and edit the page setups. Page Setups make it easy to control how different layouts are plotted, instead of having to change the settings each time you plot. To display the Page Setup Manager dialog box, use the PAGESETUP command or choose File→Page Setup Manager. To find out more about page setups, display the Page Setup Manager dialog box and click the Help button.

To create a new layout in a drawing, you can create one from scratch, copy an existing one, use a wizard, or import one from an existing drawing file. If you have Drawing Template (DWT) files that you already use with predefined layers, dimension styles, text styles, and other formatting settings, consider creating a DWT file that contains all your formatting and layouts or just your layouts so you can import them into a drawing as needed.

When you import a layout, you import the page setup settings and graphical objects that are on that layout. The LAYOUT command creates a layout from scratch or imports one from an existing file, while the LAYOUTWIZARD command allows you to create a layout using the Layout Wizard. Choose Insert→Layout and then select one of these options.

---

### Exercise 20-2: Creating a New Layout by Using the Layout Wizard

For this exercise, open the drawing file `Exercise20-2.dwg` from the Chapter 20 folder in the My Practice Drawings folder under My Documents. Then practice creating a layout by using the Layout Wizard.

1. Choose Format→Layer to display the Layer Properties Manager dialog box.

2. In the Layer Properties Manager dialog box, click the sun icon under the New VP Freeze column for the layers 3Dconst, Dims, Hidden, and Object. Double-click the VPORTS layer to set it current. Click OK to close the dialog box.

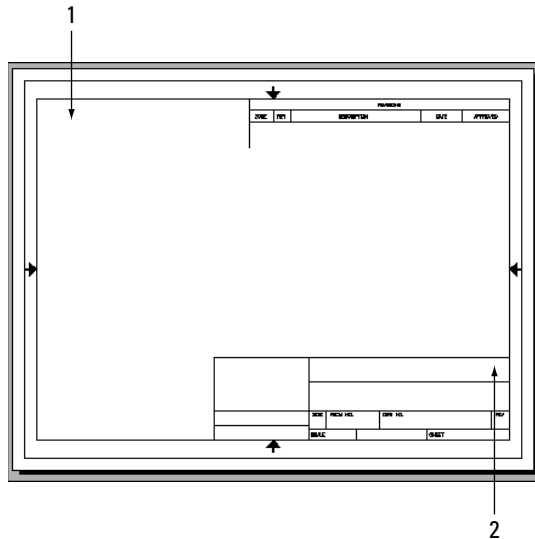
Each layer is frozen in the new viewports that you're creating.

3. Choose Insert→Layout→Create Layout Wizard.
4. In the Create Layout Wizard, replace the default layout name Layout1 with **3D Views** in the Name text box and click Next on the Begin page.
5. On the Printer page, select None from the Select a Configured Plotter for the New Layout list and click Next.
6. On the Paper Size page, select Inches under the Drawing Units section and then Letter (8.50 x 11.00 Inches) from the Paper Size drop-down list. Click Next.
7. On the Orientation page, select Landscape and click Next.
8. On the Title Block page, select ANSI A title block.dwg from the Title Block list and Block from under the Type section. Click Next.



If ANSI A title block.dwg isn't listed in the Title Block list, select None or a different title block that you have available instead.

9. On the Define Viewports page, select Std. 3D Engineering Views from the Viewport Setup section and select Fit to Scale from the Viewport Scale drop-down list. Click Next.
10. On the Pick Location page, click Select Location and pick the points 1 and 2 indicated in Figure 20-1 in order to define the sizes of the viewports. Click Next.

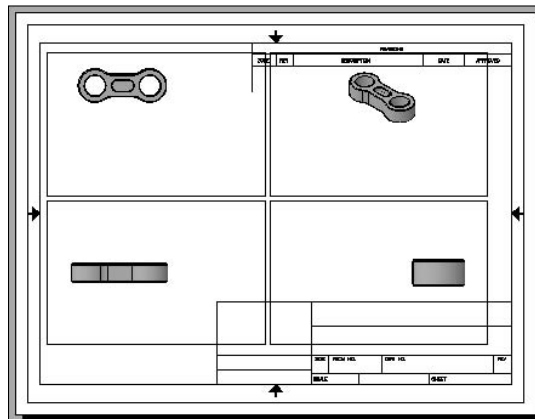


**Figure 20-1:**  
Defining the  
location  
for the  
viewports.

11. On the Finish page, click Finish.

The viewports are created for you. They look similar to those in Figure 20-2.

12. Save the drawing with the name Completed20-2.dwg to the Chapter 20 folder in the My Practice Drawings folder under My Documents and close the drawing.



**Figure 20-2:** The desired results after resizing and adjusting the viewports that were created.

Model courtesy of Mark Hechel of MasterGraphics, Inc.

## Floating viewports

*Floating viewports* are the portals (or *windows*) into Model Space that allow you to see your model from a layout. AutoCAD usually creates a single floating viewport, but you can have more than one viewport on a layout (refer to Exercise 20-2) when you use the Layout Wizard to create a new layout. To create floating viewports, you use the `-VPOR`s command or choose `View > Viewports` and select one of the options when a layout is active.

The `-VPOR`s command allows you to do the following:

- ✓ Turn a viewport on or off to display the objects from Model Space in it.
- ✓ Create between one to four viewports based on a specified area and an arrangement of how the area should be divided up.
- ✓ Control the visual style that a viewport should use to display the objects from Model Space with.
- ✓ Lock a viewport so you don't accidentally pan or zoom in the viewport, resulting in the view changing.
- ✓ You can clip a floating viewport so you see only a portion of the objects from Model Space. This causes the viewport to appear irregular in shape instead of rectangular like it does by default.
- ✓ Create an irregular-shaped viewport using a closed object such as a circle, ellipse, or 2D polyline.
- ✓ Create floating viewports based on a named configuration of tiled viewports created on the Model tab.
- ✓ You can reset the layer property overrides for selected viewports to their default values.

After viewports are created, you can modify them by using grips, the Properties palette, and the Layer Properties Manager dialog box. Grips are handy to resize a floating viewport, but the Properties palette allows you to lock a floating viewport and adjust the scale to display the objects from Model Space, set the visual style to use during plotting, and change a few other settings. The Layer Properties Manager allows you to set property overrides for layers that are displayed in the viewport and freeze layers so they appear in one viewport but not in another, allowing you to control the visibility of 2D and 3D objects based on the layers that they are on.

To find out more about creating and modifying floating viewports, see the topic *Create and Modify Layout Viewports* in AutoCAD's Online Help system. To find the topic, click the *Contents* tab and then choose *User's Guide* → *Choose a Work Process Before You Begin* → *Create Multiple-View Drawing Layouts (Paper Space)*.

### Exercise 20-3: Creating a Single Viewport and Modifying Its Properties

For this exercise, open the drawing file `Exercise20-3.dwg` from the Chapter 20 folder in the My Practice Drawings folder under My Documents. Then practice adjusting viewports.



1. Select the four viewports and display the Properties palette. From the Properties palette, select 1:1 from the Standard Scale property to set the scale for the objects in the viewports.

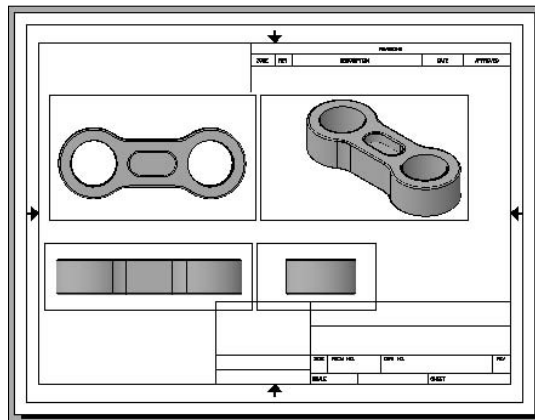
You can also adjust the scale of a viewport by selecting the viewport and then using the VP Scale list on the application or drawing status bar.

2. Double-click inside the border of the viewport in the upper-left to enter Model Space. In Model Space, pan until the 3D model is centered.

Be sure not to zoom in the viewport; otherwise, you have to set the scale back to 1:1.

3. Click in the other viewports and center the 3D model in those, too.
4. Double-click outside the viewport to return to Paper Space.
5. Select one of the viewports and adjust its size by using grips.

Adjust all of the viewports so they look like those in Figure 20-3.



**Figure 20-3:** The scale is adjusted in the viewports, and the viewports are resized.

*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

6. Select each viewport and right-click. From the shortcut menu, choose *Display Locked* → *Yes*.

Locking the display of a viewport keeps you from accidentally changing the view within a viewport.

7. Save the drawing with the name `Completed20-3.dwg` to the Chapter 20 folder in the My Practice Drawings folder under My Documents and close the drawing.



2. Exercise 20-3 shows how to set the viewports' scale by using the Standard Scale property from the Properties palette. How can you adjust the list of scales?
- A. Add a new scale.
  - B. Delete a scale.
  - C. Reorder the scale list.
  - D. None of the above.
  - E. All of the above.

## Plotting and Publishing

Drawing files from AutoCAD are plotted, not printed. Hardcopies are created using a device called a *plotter* (printer). In the past, plotters were similar to an inkjet printer, as they utilized ink cartridges or pens. These days, plotters are much more like a laser jet printer because they use toner, which is much more efficient than ink cartridges. Both the plotters of today and past use rolls of paper in various widths instead of trays of paper. AutoCAD offers two different features that have the same goal in mind: plotting and publishing (outputting to an electronic file and creating a hardcopy for sharing with others, respectively).

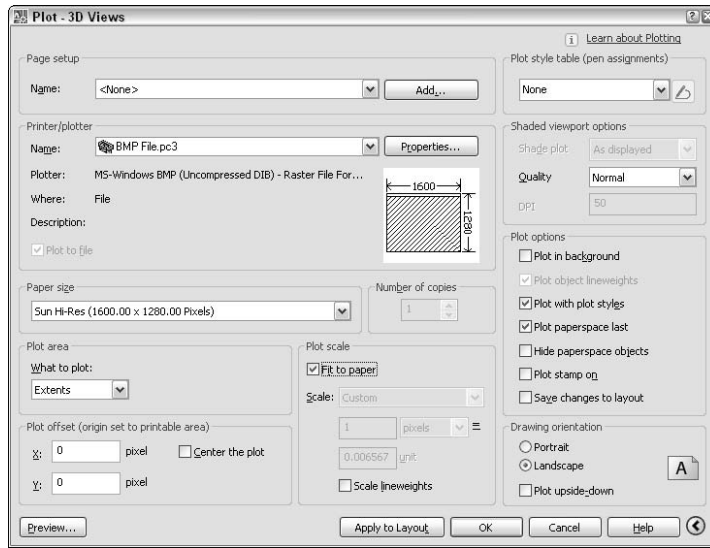
### Plotting

You can create an electronic representation of the objects in a drawing or create a hardcopy of a drawing. To do so, you use the PLOT command (choose File→Plot). You typically plot a drawing to share it with others who don't have access to the design files. When you plot a drawing file, you designate the device to use to create the output. You select the device from the Name drop-down list under the Printer/Plotter section of the Plot dialog box. The devices that you can choose are either system printers (Windows) or non-system printers (AutoCAD and the Add Plotter Wizard). Non-system printers are defined by PC3 files, which I cover under the section "Plotters," earlier in this chapter.

Most of the options in the Plot dialog box (see Figure 20-4) are identical to those in the Page Setup dialog box. If you didn't create and apply a page setup when you created the drawing, you can set the properties in the Plot dialog box or select a page setup from the Name drop-down list under the Page Setup section. The main settings that you need to set at a minimum when plotting are the device you're plotting to; paper size; plot area (if you're plotting a layout, set this to Layout); and plot scale (which should be set to 1:1 when plotting a layout).

Before you plot a drawing, make sure that the floating viewports on a layout are configured correctly. You need to check that the correct layers are displayed, set the correct scale for the Model Space objects to be plotted at, and set the Shade Plot mode that you want the Model Space objects to be plotted with. When plotting from the Model tab, you set the same options at a minimum as you do with plotting from a layout tab except you set the Shade Plot option and the plot scale to the size that the drawing is plotted at.





**Figure 20-4:**  
The Plot  
dialog box.



3. Before you can plot a drawing, you have to configure the floating viewports. How do you configure viewports from the Properties palette, Model tab, and the Layer Properties dialog box?

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To find out more about the settings in the Plot dialog box, display the Plot dialog box and click the Help button located in the lower-right corner to display AutoCAD's Online Help system.

### Exercise 20-4: Plotting a Drawing to a Raster Image File

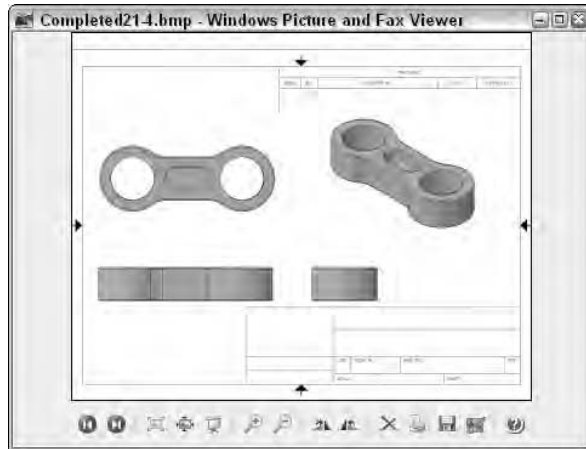


For this exercise, open the drawing file `Exercise20-4.dwg` from the Chapter 20 folder in the My Practice Drawings folder under My Documents. Then practice plotting to a BMP file.

1. Choose `File` → `Plot` to display the Plot dialog box.
2. In the Plot dialog box, select `BMP File.pc3` from the Name drop-down list under the Printer/Plotter section.  
If `BMP File.pc3` isn't listed in the drop-down list, go back and do Exercise 20-1.
3. In the Paper Size Not Found message box, select User default paper size and click `OK`.
4. Select `Extents` from the What to Plot drop-down list under the Plot Area.
5. Check `Fit to Paper` under the Plot Scale area.
6. Click `OK` in the Plot dialog box.
7. In the Browse to Plot File dialog box, browse to the Chapter 20 folder in the My Practice Drawings folder under My Documents.

8. In the File Name text box, replace the default value with **Completed20-4** and click Save.
9. Browse to the Chapter 20 folder in the My Practice Drawings folder under My Documents and double-click the Completed20-4 .bmp file to view it in your default image viewer.

The image looks similar to Figure 20-5.



**Figure 20-5:**  
A BMP file  
created  
by using  
the BMP  
File.pc3 file.

*Model courtesy of Mark Hechel of MasterGraphics, Inc.*

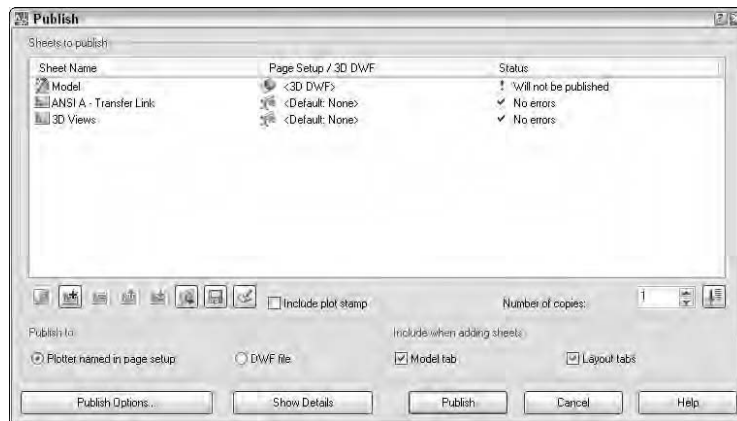
10. Close the drawing file without saving any changes to it.

## Publishing

The PLOT command outputs your drawings to either an electronic file or a hardcopy, but you're limited to plotting one layout at a time, which can be very time consuming if you need to plot more than a few layouts. The PUBLISH command (choose File⇨ Publish) allows you to plot multiple layouts across multiple drawings very quickly. You can create both electronic files and hardcopies by using the PUBLISH command, which allows you to plot all layouts and Model tabs of drawings with the page setup they were defined with, plot them all to a DWF file, or specify the creation of 3D DWF files.

The PUBLISH command displays the Publish dialog box (see Figure 20-6). From the Publish dialog box, you can add or remove sheets (or layouts) from drawings. You also have the ability to control the order in which the layouts are published and the number of times they're published, which is great if you need to create multiple hardcopies of your drawings. You then have the option to save the list of layouts so you can republish them if you need to later on (even if you've made changes to the drawings).

To find out more about publishing drawings with the Publish dialog box, display the Publish dialog box and click the Help button located in the lower-right corner to display AutoCAD's Online Help system.



**Figure 20-6:**  
The Publish  
dialog box.

## Publishing Your Models to the Web

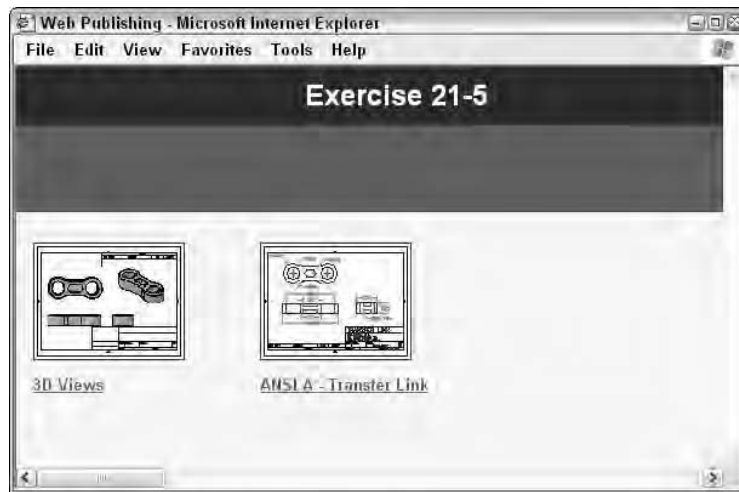
Delivering designs to your clients faster and in a way that multiple people can access them at one time ensures that you complete a project on time — maybe even at or under budget. AutoCAD supports connectivity with its project collaboration site called Buzzsaw ([www.autodesk.com/buzzsaw](http://www.autodesk.com/buzzsaw)). Buzzsaw does require you to pay a monthly fee but helps you stay in communication with your team members from start to finish on a project. Buzzsaw supports the ability to keep track of file revisions and drawing markups using Autodesk Design Review.

Buzzsaw is more geared towards architectural projects, but it works for manufacturing projects as well. For the manufacturing community, Autodesk offers Streamline ([www.autodesk.com/streamline](http://www.autodesk.com/streamline)), which is similar to Buzzsaw. Other companies offer Web-based project management and collaboration, such as Atrium (<http://atrium.pragmatics.com>) and VCollab ([www.virtuale3d.com](http://www.virtuale3d.com)).

In addition to project collaboration and management products, you can post files to an FTP site for a client to download and view or create an extranet that can display images and information that are specific to a project. An *extranet* is a Web site that is set up for clients to access and view the progress of their projects; the site is commonly protected by a password to make sure only authorized people have access to it.

You can use the Publish to Web feature to generate a Web page and a series of image files for a client to review for a project. To start the Publish to Web feature, use the PUBLISHTOWEB command (choose File→Publish to Web). You have the option to create a new Web page or work with an existing one. The Publish to Web feature creates files with the PTW file extension. Then enter a description for the Web page, select the image type to use (PNG, JPG, or DWF), and specify a template that defines how the Web page and the content generated will appear.

You then select a theme to apply and pick layouts from the drawing files to add to the Web page. Because the process is automated through a wizard, you don't have to worry about knowing how to create Web pages yourself, but you might need a little assistance from your IT department in getting the files uploaded onto the company Web site. Figure 20-7 shows an example of a Web page that was created from two layouts with the Publish to Web dialog box.



**Figure 20-7:**  
A Web page  
generated  
with the  
Publish to  
Web  
feature.

### Exercise 20-5: Creating a Web Page of Your Designs with Publish to Web

For this exercise, open the drawing file `Exercise20-5.dwg` from the Chapter 20 folder in the My Practice Drawings folder under My Documents. Then practice creating a Web page using the Publish to Web feature.

1. Choose **File** → **Publish to Web**.
2. In the Publish to Web dialog box, select **Create New Web Page** and click **Next** on the **Begin** page.
3. On the **Create Web Page** page, enter **Exercise 20-5** in the **Name of Your Web Page** text box.
4. Click the **Ellipsis** button to the right of the **Parent Directory** text box.
5. In the **Select Directory to Place Web Page** dialog box, browse to the Chapter 20 folder in the My Practice Drawings folder under My Documents and click **Open**.
6. Click **Next**.
7. On the **Select Image Type** page, select **PNG** from the **Image Type** drop-down list and select **Medium** from the **Image Size** drop-down list. Click **Next**.
8. On the **Select Template** page, select **Array of Thumbnails** from the **Templates** list and click **Next**.
9. On the **Apply Theme** page, select **Classic** from the **Themes** drop-down list and click **Next**.
10. On the **Enable i-drop** page, click **Next**.  
The i-drop feature allows you to drag and drop drawing files from the Internet or intranet into your drawing files.
11. On the **Select Drawings** page, select **3D Views** from the **Layout** drop-down list and click **Add**. Add the **ANSI A – Transfer Link** layout as well. Click **Next** after both layouts have been added.
12. On the **Generate Images** page, select **Regenerate Images for Drawings that have Changed** and click **Next**.

You would select **Regenerate All Images** if you wanted to republish all the selected layouts.

13. On the Preview and Post page, click Preview to launch your browser with the Web page.

The created Web page looks like the one in Figure 20-7.

14. Close your Web browser by clicking the X in the upper-right corner.
15. Click Finish in the Publish to Web dialog box.
16. Close the drawing file without saving any changes to it.

## Answers to Practice Opportunities

1. **To configure a plotter to create a DBX file, select AutoCAD DXB file from the Manufacturers list box.**

Remember to enter a name for the plotter in the Plotter Name text box.

2. **How can you adjust the list of scales?**

E. You can adjust the scales by choosing Format↔Scale List. In the Edit Scale List dialog box, you can do any of the following:

- **Add a new scale.** Click the Add button. Enter the name of the scale and a value to specify the units for the scale.
- **Reorder the scale list.** Select a scale and click Move Up or Move Down.
- **Delete a scale.** Select a scale, and click the Delete button.

You can also do these:

- **Reset the scale list.** Click Reset to revert the items in the scale list to the defaults that AutoCAD shipped with.
- **Edit a scale.** Select a scale and click the Edit button.

3. **How do you configure viewports from the Properties palette, Model tab, and the Layer Properties dialog box?**

- **From the Properties palette:** Select a viewport from a layout tab and adjust the properties under the Misc category. You can adjust these properties: how objects display, if the viewport is locked, and the visual style that displays the objects.
- **From the Model tab:** Restore a named view to control which part of the model is displayed. By using a named view, you can display a background in the viewport as well if you plan to use a render preset.
- **From the Layer Properties Manager dialog box:** Select which layers you want to freeze or thaw based on the viewport, apply layer, linetype, color, lineweight, and plot style.



## Chapter 21

# Importing and Exporting 3D Models

### *In This Chapter*

- ▶ Understanding model exchange formats
- ▶ Working with file translators
- ▶ Using 3D models with non-CAD applications

**I**f you're working in a team environment, not everyone will have access to AutoCAD — nor would you want everyone on the design team to have access to design files. Because a design team can consist of people in sales and marketing at times, as well as outside vendors, you need to understand the file formats that you can use to exchange files. AutoCAD offers a variety of file formats that you can export from and import into AutoCAD. If AutoCAD doesn't offer a file format you can use, external file translators allow you to convert a file from one format to another. You can export 3D models to a variety of file formats that support 2D/3D information or 2D information only.

## *Exchanging 3D Models*

AutoCAD supports the ability to exchange 15 different file formats. Ten of the file formats that AutoCAD supports for exchange are vector file formats, and the other five are raster file formats (image files).



You can't import or export all of the file formats that AutoCAD supports for exchange from AutoCAD.

### *Vector file formats*

This section covers the vector file formats that you can export and import into AutoCAD to exchange 3D models. Vector files store model information as objects such as lines, arcs, surfaces, 3D solids, and so on, which makes it possible to modify and edit the geometry. Vector files are usually smaller than raster files.

#### **DWG**

The drawing file format (.dwg) is native to AutoCAD, and AutoCAD uses it to store 3D models. You can use this file format to exchange files with users of Autodesk products and other applications that can exchange DWG files. When exchanging AutoCAD 2007 or 2008 files with users of older versions of AutoCAD, select the correct file format from the Files of Type drop-down list in the Save As dialog box. Some of the 3D solid and surface objects didn't exist in releases prior to AutoCAD 2007, so the objects appear correctly but aren't available for edit. You also might have problems with lights and materials when saving to an older release of AutoCAD or opening a drawing created in AutoCAD 2008 in AutoCAD 2007.

***DXF***

The Drawing Interchange Format (.dxf) is the official exchange file format that Autodesk supports for exchanging drawing files with AutoCAD and other CAD applications. DXF files can contain both 2D and 3D objects like DWG files can. Unlike the DWG file format, the DXF file format is a well-documented open file format that allows other CAD software vendors to incorporate the file format into their own applications. You can create DXF files by using the Save As dialog box and selecting one of the available DXF file formats from the Files of Type drop-down list. You can open a DXF file by using the OPEN command (choose File→Open) and selecting DXF (\*.dxf) from the Files of Type drop-down list in the Open dialog box.

***DGN***

Microstation Design V8 (.dgn) is one of the latest additions to the file formats that AutoCAD supports. DGN files are created by the products developed by Bentley Systems, Inc., a competitor of Autodesk. DGN files can contain both 2D and 3D information just like DWG files can. To export a DWG file to DGN, you use the DGNEXPORT command; choose File→Export and select V8 DGN (\*.dgn) from the Files of Type drop-down list. To import a DGN file, you use the DGNIMPORT command; choose File→Import and select V8 DGN (\*.dgn) from the Files of Type drop-down list. If you want to attach a DGN file, you can use the DGNATTACH command (choose Insert→DGN Underlay) or the External References palette (choose Insert→External References).

***SAT***

ACIS solid (.sat) is a file format that contains 3D geometry that you can exchange with a number of 3D applications. You can think of SAT files as the 3D equivalent of DXF files. To export a SAT file, you use the EXPORT command; choose File→Export, and select ACIS (\*.sat) from the Files of Type drop-down list. AutoCAD does support the ability to import a SAT file into a drawing by using the commands ACISIN or IMPORT; choose File→Import, and select ACIS (\*.sat) from the Files of Type drop-down list.

***STL***

Solid Object Stereolithography (.stl) is another standardized 3D file format that you can use to exchange 3D models with other CAD applications and 3D visualization applications. To export a STL file, you use the EXPORT command; choose File→Export and select Lithography (\*.stl) from the Files of Type drop-down list. AutoCAD doesn't support the ability to import a STL file, but file translators on the market do allow you to convert a STL file into a SAT file. (I cover file translators later in this chapter.)

***WMF***

The Windows Metafiles format (.wmf) allows you to exchange 2D representations of a 3D model using very basic geometry types like text, lines, and arcs that might make up more complex objects like polylines or 3D solids. WMF files are commonly used in graphic arts programs as a basic vector-based file format that allows you to exchange files between Adobe Illustrator, Corel Draw, and AutoCAD, among others, with ease.

To export a WMF file, you use the WMFOUT command; choose File→Export, and select Metafile (\*.wmf) from the Files of Type drop-down list. The WMFOPTS command gives you a few options to control how a WMF file is created when it is exported. AutoCAD does support the ability to import a WMF file into a drawing by using the commands WMFIN or IMPORT; choose File→Import, and select Metafile (\*.wmf) from the Files of Type drop-down list. The system variables WMFBKGND and WMFFOREGND control the background and foreground color assignments when inserting a WMF file into AutoCAD.

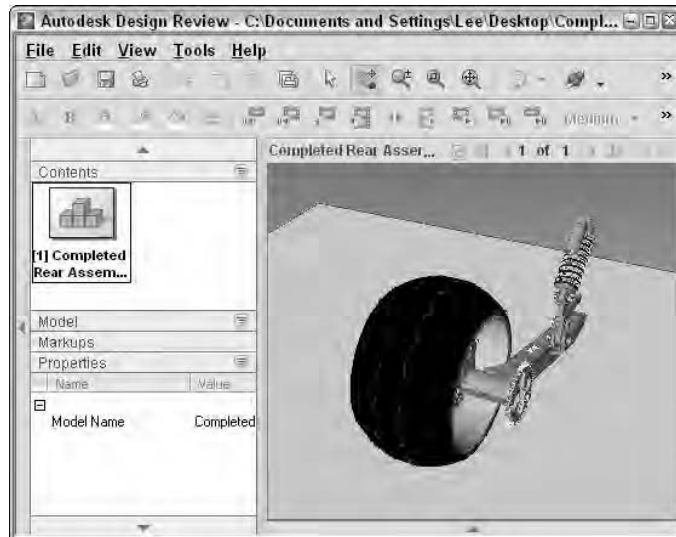


## DWF

Design Web Format (.dwf) files are a creation of Autodesk, and you use them for exchanging 2D and 3D designs with non-CAD users. DWF files are an electronic representation of a plotted drawing. 2D DWF files can contain a single layout or multiple layouts, whereas a 3D DWF file doesn't contain any layouts but can contain a hierarchical view of a 3D model and can only contain 3D objects.

To create a 2D DWF file, you use either the PLOT command (choose File→Plot) or the PUBLISH command (choose File→Publish). 3D DWF files are created with the 3DDWF command; choose File→Export, and select 3D DWF (\*.dwf) from the Files of Type drop-down list or use the PUBLISH command. When creating a 3D DWF file, you have some control over how the file is generated by choosing Tools→Options in the Export 3D DWF file dialog box. The system variable 3DDWFPPREC also contributes to how accurate a 3D DWF file appears. For more information on the settings for the 3DDWFPPREC system variable, enter **3ddwfp** at the command prompt and press F1 to open the topic page for the system variable in AutoCAD's Online Help system.

You attach 2D DWF files to a drawing in a similar fashion to the way you attach DWG and DGN files. To attach a 2D DWF file, you use the DWFATTACH command (choose insert→DWF Underlay). Anyone can view DWF files via a free viewer, which makes it a good format when you're sharing files with non-AutoCAD users. The free viewer is called Autodesk Design Review ([www.autodesk.com/designreview](http://www.autodesk.com/designreview)). Autodesk Design Review allows you to view and print DWF files, along with creating markups and performing measurements on design files. If you have Windows Vista, you can view and print the new DWFx files directly without the viewer. Figure 21-1 shows a 3D DWF file opened in Autodesk Design Review.



**Figure 21-1:**  
A 3D DWF  
file of a  
custom rear  
assembly  
for a  
motorcycle.

Model courtesy of Mark Hechel of MasterGraphics, Inc.



Don't confuse Autodesk Design Review with the program Autodesk DWF Viewer. Autodesk DWF Viewer is limited to being able to view and print only DWF files; this is the version of the viewer prior to the release of Autodesk Design Review.

## EPS

Encapsulated Postscript (.eps) files are used primarily with publishing and graphic arts applications such as Corel Draw and Adobe Illustrator. You can use EPS files to create exploded views for assembly documents or other types of documents that might show

line art drawings. To create an EPS file, you use the PSOUT command; choose File⇨ Export and select Encapsulated PS (\*.eps) from the Files of Type drop-down list. You can control a number of settings when you create an EPS file by choosing Tools⇨ Options to display the PostScript Out Options dialog box while in the Export Data dialog box.

### ***PLT***

AutoCAD Plot (.plt) files are generated when you plot a drawing to a file instead of a plotter. PLT files contain only a 2D representation of the plotted drawing. You can create PLT files to send to a reprographics company that specializes in plotting large volumes of drawings. To create a PLT file, select the Plot to File option in the Plot dialog box. A PLT file is similar to a PRN file that you can print to a Windows system printer. PLT files contain some additional instructions for the plotter to plot the drawing correctly.

## ***Raster file formats***

This section covers the raster file formats that you can export out of AutoCAD to exchange data from 3D models with non-CAD users. Raster files don't maintain the ability to modify an object like a line or an arc, and they store information as a series of colored pixels, which causes the information to be converted into 2D data instead of remaining 3D data. Raster images are typically larger files than vector files.

You can attach all the raster image file formats — with the exception of PDF files — to a 3D object using the Texture Map map type. Raster images can also be used as a background of the current viewport during rendering or attached to a drawing with the IMAGEATTACH command (choose Insert⇨Raster Image Reference). The raster images (not excluding PDF files) can also be created from a rendered image that is generated in the Render Window using the Save option.

### ***BMP***

Windows Bitmap (.bmp) files are one of the most common image files that are exchanged with non-CAD users. The downside to BMP files is that they don't offer any built-in file compression, which makes them very large and hard to send through e-mail. You can use just about any image viewer or Web browser to view a BMP file, which make them nice for exchanging with your design team. Use the BMPOUT command, which allows you to select which objects in the current view you want to export, when you create the BMP file. Or use the EXPORT command; choose File⇨ Export and select Bitmap (\*.bmp) from the Files of Type drop-down list.

### ***TIF***

Tag Image File Format (.tif) files aren't everyday image files used for exchanging 3D models because they are very large. Your marketing department may request TIF files though, because TIFs are good for creating brochures for sales literature or pricing catalogs. TIF files are not commonly sent through the Internet due to their file size. You create TIF files with the TIFOUT command, which allows you to select the objects in the current view that you want to export.

### ***PNG***

Portable Network Graphics (.png) files are one of the most commonly used image files for exchanging images or to post on a company Web site because they offer the same quality as a BMP file but use compression technology that makes them a fraction of the file size; they're optimal for e-mailing and using on the Internet. You can create PNG files with the PNGOUT command, which allows you to select the objects in the current view that you want to export. In addition to the PNGOUT command, you can plot to a PNG file by using the PublishToWeb PNG.pc3 plotter.

**JPG**

JPEG (.jpg) graphic files are all-around graphic files that offer compression and the ability to control the file quality, which can increase or decrease the file size. JPEG files — at about the same size as a PNG file — are usually much smaller than BMP files so they can be transmitted through e-mail and used on the Internet. You can create JPEG files with the JPGOUT command, which allows you to select the objects in the current view that you want to export. In addition to the JPGOUT command, you can plot to a JPG file by using the PublishToWeb JPG.pc3 plotter.

**PDF**

Adobe Acrobat Portable Document (.pdf) files are often associated with Word documents, electronic brochures, and other office documents such as standard forms. PDF files can contain 2D and 3D information, but AutoCAD can generate only a PDF file that contains a 2D representation of your 3D model. To create a 3D representation of your 3D model, you need to purchase a copy of Adobe Acrobat 3D. To find out more information about Adobe Acrobat 3D, visit Adobe's Web site ([www.adobe.com/products/acrobat3d](http://www.adobe.com/products/acrobat3d)).

Viewing PDF files requires a special viewer called Adobe Acrobat Reader, which is free and is installed on most computers these days. If you don't have Adobe Acrobat Reader installed on your computer, you can download it from Adobe's Web site ([www.adobe.com](http://www.adobe.com)). AutoCAD doesn't support the ability to import or attach PDF files like it does for other raster file formats. To create a PDF file in AutoCAD, use the Plot dialog box and the plotter named DWG to PDF.pc3.

## File Translators

Because the economy has gone global since the turn of the century and the introduction of the Internet, *file translators* have become essential in the way businesses operate. A file translator is an application that allows you to take an electronic file of a specific format and convert it into a different file format. Using a translator makes it easier to convert files with users who might use a different release of AutoCAD or to a different CAD file format altogether. Companies like Autodesk build in the most common file formats that most customers want, but at times that isn't enough. File translators are often developed by third parties that obtain the rights to use a file format from the company that publishes the file or dissect a file format and create an application that can read and write a file from one format to another, allowing files to be opened in different applications with greater ease.



Some of the file translators on the market are free, and some you must purchase. Although a free deal is hard to pass up, you often get what you pay for.

File translators aren't the end-all solution, but they can eliminate the need to purchase software that you might not really use every day. Take a look at these translators to help exchange 3D models:

- ✓ **TransMagic** ([www.transmagic.com](http://www.transmagic.com)): The TransMagic products allow you to convert AutoCAD files from a number of popular file formats, such as IGES, STEP, STL, Pro/ENGINEER, ACIS, and Inventor, among others. They translate 3D models created in AutoCAD using either the STL or ACIS file format.
- ✓ **CADlook** ([www.cadlook.com](http://www.cadlook.com)): The CADlook Batch Translators allow you to translate SAT/ACIS files to IGES, STEP, VDA, X\_T, VRML, and STL.

## Harnessing 3D Models in Non-CAD Applications

It's great to kick back in the cubicle (or maybe even the corner office with a window) and enjoy your 3D models in AutoCAD, but the reality is that at some point your 3D model must leave the confines in which it was created. When the 3D model does finally leave the nest (also known as AutoCAD), it most likely appears in a presentation shown to a client or whisked across the Internet in an e-mail. You can import a raster file directly into a document that you create in Microsoft Word or PowerPoint, but you can also embed a lightweight representation of your 3D model in the form of a DWF file into a Microsoft Word document or PowerPoint presentation.

DWF files are embedded in a Word document or PowerPoint presentation with little effort, and they can add a little extra glitz and dazzle for the client. When you embed a DWF file in an application, you create a link to an installed component on the computer that displays a file created in another application. In this case, the installed component is called the Autodesk DWF Viewer Control.

To use the Autodesk DWF Viewer Control, the Autodesk Design Review or DWF Viewer needs to be installed on the computer in order to view the file. The Autodesk DWF Viewer Control allows you to zoom, pan, and orbit directly in the file that the control is embedded in, which can make a presentation more interesting for the client to watch.



### Exercise 21-1: Creating a 3D DWF File

For this exercise, open the drawing file `Exercise21-1.dwg` from the Chapter 21 folder in the My Practice Drawings folder under My Documents. Then practice creating a 3D DWF file.

1. At the command prompt, enter `3ddwfpref` and press Enter.
2. At the Enter new value for `3DDWFPPREC <2>`: prompt, enter `4` and press Enter.  
A value of 4 is a greater precision value to create the 3D DWF and makes the drawing appear smoother than the default value of 2.
3. Choose `File`→`Export`.
4. In the Export Data dialog box, select 3D DWF (\*.dwf) from the Files of Type dropdown list.
5. Choose `Tools`→`Options`.
6. In the 3D DWF Publish dialog box (see Figure 21-2), make sure that the All Model Space Objects option is selected and the Publish with Materials check box is selected. Click OK.
7. Browse to the Chapter 21 folder in the My Practice Drawings folder and enter the name **Completed21-1** in the File Name text box. Click Save.
8. When the message box appears, click Yes to view the 3D DWF file created with Autodesk Design review (or Autodesk DWF Viewer based on what is installed) or No to not view the DWF file created.
9. Close the viewer when you're done reviewing the model.  
Many of the navigation tools in the viewer are similar to those in AutoCAD. The 3D DWF file in the viewer looks similar to the one shown earlier in Figure 21-1.
10. Close the drawing and don't save any changes.



**Figure 21-2:**  
The 3D DWF  
Publish  
dialog box.

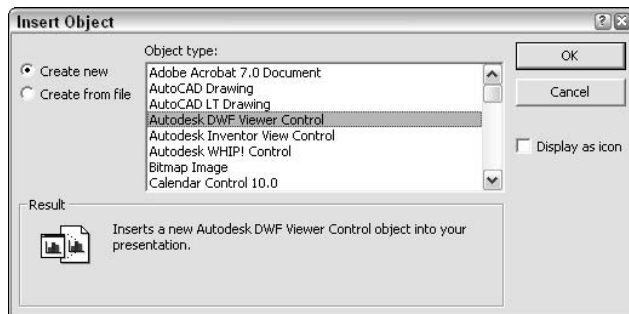


### Exercise 21-2: Embedding a 3D DWF File into PowerPoint

In this exercise, you practice embedding a 3D DWF file into PowerPoint.

- To launch PowerPoint 2003, choose Start→(All) Programs→Microsoft Office→Microsoft Office PowerPoint 2003.  
To launch PowerPoint 2002, choose Start→(All) Programs→Microsoft PowerPoint.  
The location of the shortcut to start PowerPoint varies based on the version of Microsoft Office you have.
- In PowerPoint, choose Insert→New Slide and then delete all the objects off the slide by selecting them and pressing the Delete key.
- Choose Insert→Object.
- In the Insert Object dialog box (see Figure 21-3), select Autodesk DWF Viewer Control and click OK.

A new object is added to the slide. You can select the object and use the grips along the edges to resize the object if you need to.

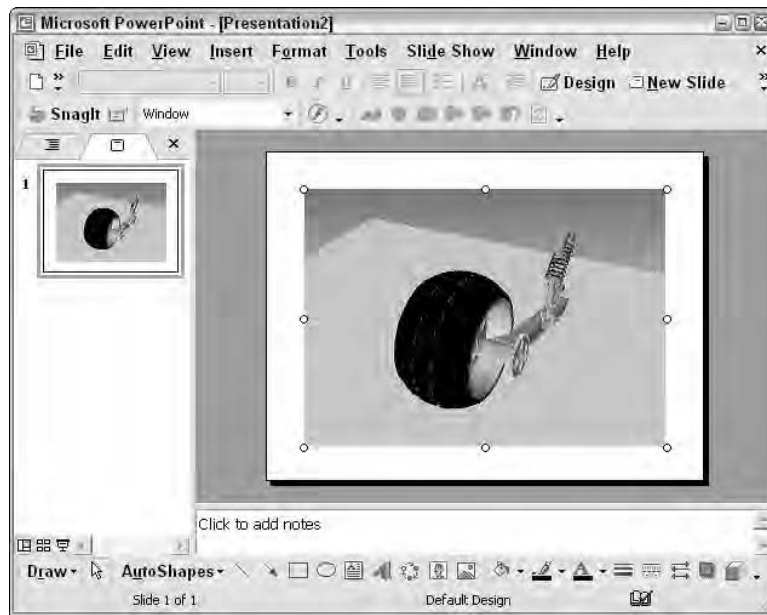


**Figure 21-3:**  
The Insert  
Object  
dialog box.

- Right-click the object that you just inserted and choose Autodesk DWF Viewer Control Object→Properties from the shortcut menu.
- In the Autodesk DWF Viewer Control Properties dialog box, click Browse.
- In the Path to the File You Wish to View dialog box, browse to the Chapter 21 folder in the My Practice Drawings folder and select Completed21-1.dwf. Click open.

The Completed21-1.dwf file was created in Exercise 21-1. If you don't have the file, complete Exercise 21-1 first.

8. In the Autodesk DWF Viewer Control Properties dialog box, click OK.  
The PowerPoint presentation looks like the one shown in Figure 21-4.
9. Choose Slide Show→View Show or press F5.  
The slide show starts to run. The embedded 3D DWF file has the same set of toolbars and controls that were displayed in Autodesk Design Review (or Autodesk DWF Viewer).
10. Save the PowerPoint presentation with the name `Completed21-2.ppt` to the Chapter 21 folder in the My Practice Drawings folder under My Documents. Then close the drawing.



**Figure 21-4:**  
A  
PowerPoint  
presentation  
with an  
embedded  
3D DWF file.

Model courtesy of Mark Hechel of MasterGraphics, Inc.

# Part VII

# The Part of Tens

The 5th Wave

By Rich Tennant



## *In this part . . .*

**A**utoCAD is a very complex application for new users to start with, and because it changes so quickly from release to release, many features are often overlooked. This part gives you an overview of the new features introduced in AutoCAD 2008 that you can use to improve your skill set and drafting efficiency. Although the Online Help system that comes with AutoCAD contains a lot of documentation and is very complete, it also contains a lot of information, making it hard at times to find just what you are looking for. So in this part, I provide a listing of a variety of Web sites that you can visit to find information on features in AutoCAD and industry-specific information. Whether you're trying to find out how to create a rendering or trying to find a 3D model of a standard bolt, you should be able to find a resource to help you out.



## Chapter 22

# More than Ten New Features in AutoCAD 2008

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**E**ach year, a new release of AutoCAD delivers new features and improved productivity. This chapter brings attention to the many new features that are in AutoCAD 2008. These features can help improve productivity and the visual quality of your drawings. The content of this chapter is an overview of the many new enhancements in AutoCAD 2008, and not a full, in-depth look at each new feature and how to use it.

### *Improvements to 2D Drafting*

2D drafting has been the core focus of AutoCAD since it was first introduced in the early 1980s. With each release, Autodesk comes up with features that improve 2D drafting. AutoCAD 2007 introduced the ability to attach Design Web Format (DWF) files to a drawing as an underlay, which was similar to an external referenced drawing but in a read-only state.

With AutoCAD 2008, you now have the ability to control the visibility of individual layers of a DWF underlay as long as the layers were originally published with the DWF file. Along with being able to control the layers of a DWF underlay, other layer improvements allow you to fade the contents of a locked layer or apply property overrides to a layer for a viewport. Along with layer improvements, AutoCAD provides a new workspace for 2D drafting and allows you to work in a few different ways with design (DGN) files created with MicroStation, which is developed by Bentley Systems, Inc. ([www.bentley.com](http://www.bentley.com)).

### *New dashboard panels and workspace with a focus on 2D drafting*

The Dashboard was introduced in AutoCAD 2007 as a way to access groups of 3D features that help to improve the workflow of 3D modeling. The concept of panels and the Dashboard has been expanded to incorporate 2D drafting and annotation features, along with other general processes that are used for both 2D and 3D drafting.

The new 2D Drafting & Annotation workspace uses the Dashboard palette and a new toolbar named Standard Annotation. The Standard Annotation toolbar is a shortened version of the Standard toolbar. By default, eight control panels are displayed on the Dashboard palette when the 2D Drafting & Annotation workspace is current: Layers, 2D Draw, Annotation Scaling, Dimensions, Text, Multileaders, Tables, and 2D Navigate. To discover more about using and creating workspaces, see Chapter 3.

Table 24-1 lists the new panels that have been added to the Dashboard for 2D drafting and working with annotation in a drawing.

<i>Dashboard Panel Name</i>	<i>Description</i>
2D Navigate	Contains basic panning and zooming tools along with the ability to set a named view current.
Annotation Scaling	Contains tools that allow you to add or delete objects from the current annotation scale.
Block Attributes	Contains tools that allow you to create new blocks and to create, modify, and extract attributes in blocks.
Dimensions	Contains tools that allow you to create and modify both dimension objects and dimension styles.
Layers	Contains tools that allow you to create, modify, and manage layers, layer states, and layer filters.
Multileaders	Contains tools that allow you to create and modify both multi-leader objects and styles.
Object Properties	Contains tools that allow you to set the current entity properties that are used when creating new objects. The properties include color, linetype, lineweight, and plot style.
Tables	Contains tools that allow you to create and modify both table objects and table styles.
Text	Contains tools that allow you to create and modify single-line and multiline text objects and manage text styles. The panel also contains tools to find text and check for misspelled words.

## *Layer improvements*

AutoCAD 2008 allows you to override the color, linetype, lineweight, and plot style properties of layers by viewport. This can help make working with client drawings much easier. Instead of translating all the layers in drawings to match your standards for plotting, now you can just remap the properties of a layer for a viewport when plotting. The ability to override layer properties by viewport is also great for working with details and things like hatching where you might want the hatching to still be displayed but want it to plot lighter in one viewport.

Another new feature for layers is the ability to control the fade level of layers that are locked in the drawing. By fading locked layers, AutoCAD allows you to focus more on the editable objects in the drawing. The controls for setting the fade value of locked layers are done from the Layers panel on the Dashboard, and the fade value is applied globally across all locked layers. You can access other layer features from the new Layers panel on the Dashboard, such as isolating layers, switching to a saved layer state or filter, or changing the current layer. For more information on layers, see the topic Work with Layers in the Online Help system of AutoCAD.

## *Control layers for DWFs*

AutoCAD 2007 first introduced the ability to attach a DWF file as an underlay that was similar to attaching an external referenced drawing, but with a few limitations. When a DWF file was attached to a drawing, you couldn't control the layers that were displayed in the file, and you couldn't bind (or merge) the geometry in the DWF file with the geometry in the drawing that the DWF file was attached to. AutoCAD 2008 resolves the limitation of not being able to control which layers in the DWF file are displayed by introducing the DWFLAYERS command. The way you work with the DWF Layers dialog box is similar to the way you can work with Layers in Autodesk Design Review. For more information on working with layers in a DWF file, see the topic Control the Display of Layers in a DWF Underlay in the Online Help system of AutoCAD.

## *Work with DGN Files*

AutoCAD 2008 now allows you to import a DGN file and convert all the geometry in the file to native AutoCAD objects. DGN files are created with another popular CAD application on the market called MicroStation, which is developed by Bentley Systems, Inc. MicroStation can be found in use by the Department of Transportation (DOT) for some states, along with other companies that primarily focus on civil designs. Along with being able to import DGN files, AutoCAD can also reverse the process and take a DWG file and convert it into a DGN file.

Being able to import and export the contents of a DGN file is one way of working, but AutoCAD also allows you to attach a DGN file in mostly the same way that you can attach a DWF file. After you attach a DNG file, you can snap to its geometry much like you can snap to objects that you create in a drawing or that are part of an attached DWF file. For more information on working with DGN files, see the topics Attach DGN Files as Underlays, Import MicroStation V8 DGN Files, and Export MicroStation V8 DGN Files in the Online Help system of AutoCAD.

## *Inverse clipping for external references*

When clipping an external referenced drawing, you can now choose to invert the clipping boundary so everything inside the specified boundary is hidden instead of everything on the outside of the boundary. To invert the clipped boundary, use the Invert clip option when creating the boundary. The Invert clip option toggles the invert clipping mode on or off based on the current setting when you use the option. For more information on clipping an external reference, see the topic Clip Referenced Drawings and Blocks in the Online Help system of AutoCAD.

## *Productivity Changes to Annotating Drawings*

Annotation is probably one of the most critical things you can add to your drawings to ensure it's manufactured or built directly. Annotation comes in a variety of different formats, from tables to dimension objects and leaders. Most of the objects for annotation in a drawing have been changed slightly. The main reason for this is a new feature

called *annotation scaling*, which eliminates some problems people have had using multiple dimensions or multiple scaled blocks to show dimensions and notes at the right scale in a viewport when plotting. Many other changes also give you better control over paragraph text and improved spell checking — something that I need because of my bad spelling habits.

## ***Annotation scaling***

Annotation scaling is by far the biggest new feature that is in AutoCAD 2008 because it touches so many different features. Annotation scaling allows you to add annotation objects such as text, dimensions, hatch objects, or even blocks to a drawing in one scale but have the size/scale of the objects update automatically by changing the current annotation scale value. For a long time, many people have been placing dimensions on a layout or have been creating two sets of dimensions for the model so they could display dimensions in a uniform size across multiple viewports with different scales.

You can assign annotation scaling by viewport. Doing so turns off the visibility of the object, or it's resized in the case of hatch, text, or dimension objects. You can assign a property to blocks so they interact with annotation scaling, which can be helpful for things like grid markers, cut symbols, and notes. To use annotation scaling, you specify that the object or style that is assigned to an annotation object is designated for use with annotation scaling and then you adjust the annotation scale for the current viewport. You can adjust the annotation scale value from the status bar of the application window or the drawing window. For more information on annotation scaling, see the topic Scale Annotations in the Online Help system of AutoCAD.

## ***Multiline text improvements***

Paragraph text (or *multiline text*) has seen many improvements over the past several releases. You can find a number of significant changes to formatting paragraph text, which include being able to create columns of text, align individual paragraphs, use enhanced tabs, control indents better, and extend the use of paragraph and line spacing settings.

The new columns feature allows you to create columns of text for long specifications or notes. You can specify the columns as static numbers or have them automatically created based on the height of the multiline text object. After the number of columns is defined, you can just start typing text, and the text wraps between the columns as necessary. You can set the width of a column and even the width of the gutter that's between columns in the Column Settings dialog box. You can display the Column Settings dialog box by clicking the Columns button on the Text Formatting toolbar and then choosing Column Settings from the menu that appears.

In AutoCAD 2008, you can specify a different justification for each paragraph, and you can use many different tab options. The tab options have been improved to include center, right, and decimal alignment options in addition to the existing left tab option. Paragraph and line spacing have also been improved to give you better control of the values. To specify indents, tabs, and other paragraph formatting options, you use the Paragraph dialog box. You can display the Paragraph dialog box by clicking the Paragraph button on the Text Formatting toolbar. For more information on multiline text, see the topics Create Multiline Text and Create and Edit Columns in Multiline Text in the Online Help system of AutoCAD.

## *Changes to spell checking*

When performing spell checking, you can now check text that's part of a dimension and filter by whether you want to include words in block attributes and external references in AutoCAD 2008. Along with the ability to filter by certain types of objects, you can choose some common types of words to ignore, such as capitalized words or words with numbers.

Other improvements allow you to manage more than one custom dictionary, and you can switch between different dictionaries. The ability to manage custom dictionaries allows you to have a personal dictionary to use and a centralized dictionary that might be shared among all drafters in your department or company. When checking a drawing, you now can search the entire drawing, the current viewport/layout, or the objects you select. AutoCAD zooms to a misspelled word and highlights it to make it easier for you to identify the word in question. The SPELL command and Check Spelling dialog box (choose Tools→Spelling) check for misspelled words in a drawing. For more information on spell checking, see the topic Check Spelling in the Online Help system of AutoCAD.

## *Multiline attributes*

AutoCAD has supported the ability to create both single-line and multiline text objects for a long time, but attributes always seemed to take a backseat and were content with being single lines. It appears that attributes finally had enough and revolted, because they now support multilines as well, which is great for things like plot stamps and title blocks along with other blocks that might need to contain more than one line of information in them.

To create an attribute that supports multiple lines of text, you need to enable the Multiple Lines mode (or property) when creating a new attribute with the ATTDEF command or from the Properties palette when you want to modify existing attributes to support the new multiline behavior. When the Multiple Lines mode is enabled, the In-Place Text Editor is used to create or modify the value that's assigned to an attribute, which is like creating or editing a multiline text object. For more information on creating multiline attributes, see the topic Define Block Attributes in the Online Help system of AutoCAD.

## *Table improvements*

Tables have a lot of potential to change the way you do schedules and generate a bill of materials in a drawing. AutoCAD 2008 introduces a number of the requests that users have been asking for since tables were introduced and even before then. Some of the new changes with the way you work with the table editor will remind you of tasks that can be performed in Microsoft Excel.

The table indicator has been improved to allow you to select an entire row, column, or the whole table. You can format cells by using the new cell styles, which can be applied to an individual cell or an entire row or column. Along with cell styles, table styles have seen some improvements that give you much more flexibility in the way a table looks.

Formatting improvements aren't the only changes that are new with tables. You can now create a linked table that allows you to bring information from a Microsoft Excel spreadsheet into AutoCAD. To create a table with linked information, you use the Data Link Manager dialog box and the New Excel Data Link dialog box. You can format linked tables by default, but you can't edit the cells in linked tables unless you open the source file directly or unlock the table. If you make edits in the table, you can write the changes back to the Excel spreadsheet. To display the Data Link Manager to create a linked table, you can enter **datalink** at the command prompt or click the From a Data Link radio button in the Insert Table dialog box (choose Draw↔Table). For more information on working with tables, see the topic Create and Modify Tables or the topic Link a Table to External Data in the Online Help system of AutoCAD.

## *Dimension enhancements*

In AutoCAD 2008, dimensions see some new enhancements that help to improve the communication of your designs. One of the new features available allows you to change existing dimensions to an *inspection dimension*. It indicates critical dimensions or tolerances that need to be checked during the manufacturing of a part and how often to perform a quality assurance check. Inspection dimensions are made up of a label, the dimension value, and an inspection rate. The Inspection Dimension dialog box (choose Dimension↔Inspection) adds and removes inspection dimensions to and from existing dimensions in a drawing.

Other changes to dimensions include the ability to place a radial dimension off the object that is being dimensioned, and when this is done, an arc extension line is drawn to ensure the radius or diameter of the object is properly communicated. In addition to changes to the radial dimension, you can now specify which quadrant an angular dimension is dimensioned to, which allows you to place the dimension text outside the extension lines where a dimension line is drawn to the dimension text.

AutoCAD 2007 saw the inclusion of a jogged radial dimension to communicate the radius of an object, but the dimension used a user-defined center point instead of the true center point of the object. AutoCAD 2008 introduces a jogged *linear* dimension, but unlike the jogged radial dimension, it isn't an entirely new object; instead, it's a linear dimension with a jogged symbol added to it after the linear dimensions is placed. Jogged linear dimensions indicate when the distance between the extension lines isn't the true length of the object being dimensioned. Typically, the dimension value displayed is smaller than what the object would measure.

Some additional enhancements to dimensions are the abilities to align a tolerance by either its operational symbol or decimal separator, to space linear dimensions an equal distance apart or align parallel linear or angular dimensions, and to create breaks in a dimension where a dimension or object intersects another dimension or multileader. Adding dimension breaks to dimensions or multileaders ensures that the dimensions or leaders in the drawing aren't misinterpreted. For more information on working with dimensions, see the topic Dimensions and Tolerances in the Online Help system of AutoCAD.

## *Multileaders*

Leaders are used to call out a feature of a design, such as the number of holes on a plate if they're of a typical size or when something should be welded. Leaders serve many different purposes, and AutoCAD 2008 extends the capability of leaders by introducing a new multileader object. The multileader object, as its name suggests, can have multiple leaders to a single block or text object. By using multiple leaders, you can simplify the

editing process in the drawing if something changes. Multileaders are created with the MLEADER command. Like dimensions, you use styles to control the appearance of multileaders. You create styles for a multileader with the Multileader Style Manager (choose Format⇒Multileader Style).

Multileaders also support annotation scaling like other annotation objects do, and they can be broken with the dimension break feature. Multileaders for the most part function like leaders created with the quick leader (QLEADER) and legacy leader (LEADER) commands from previous releases. For more information on creating multileaders, see the topic Create and Modify Leaders in the Online Help system of AutoCAD.

## *Enhancements to 3D Visualization*

AutoCAD 2007 brought many much-needed improvements to visualization in AutoCAD, and AutoCAD 2008 builds on those changes with even more enhancements. Although it isn't quite as much of an evolution as it was last release, the improvements in materials and lighting and the introduction of plug-ins in this release are welcome additions to the feature set.

### *Direct 3D support*

AutoCAD has supported both hardware and software acceleration for some time now. Hardware acceleration improves the quality of the on-screen image when working with 3D modeling and visualization. With AutoCAD 2008, you can now choose to use either OpenGL or Direct 3D for the driver type when using hardware acceleration as long as your graphics card supports both types of drivers. Previous releases of AutoCAD supported only OpenGL. To find out more about hardware acceleration, see Chapter 3.

### *Backgrounds*

Backgrounds used with named views and cameras when rendering or using a 3D visual style now have the ability to generate a sun and sky. In the Adjust Sun & Sky Background dialog box, you can specify the position of the sun, its color, and other settings of the sun such as haze and the horizon. To access the Adjust Sun & Sky Background dialog box, you must enable photometric lighting in the drawing. (For more information on working with photometric lighting, see Chapter 16.) When photometric lighting is enabled, display the View Manager (choose View⇒Named Views) and select an existing view or a new view. If you select an existing view, click the Background Override property and select Sun & Sky from the drop-down list. If you create a new view, select Sun & Sky from the Background drop-down list in the New View dialog box. For more information on setting a background for a named view, see Chapter 5.

### *More lighting*

Lighting has been improved in this release to include photometric lighting, which gives the lights in a model a much more realistic behavior. Along with lights now supporting photometric settings, you have the ability to create a few additional types of lights that weren't available in AutoCAD 2007. These additional types of lights are free spotlight, target point, and weblight. For more information on setting up and using lights, see Chapter 16.

## ***Enhanced material options***

Materials give your models a more realistic look, and a lot of work has been done in this area since AutoCAD 2007. You now have access to five additional procedural maps, along with the ability to create complex materials by nesting materials. You can also designate which materials are two-sided, unlike its predecessor, which allowed you to designate two-sided materials only at a global level when rendering. Keeping with the photometric lighting changes, you can now specify the luminance of a material, which is mutually exclusive with specifying of the self-illumination attribute for a material. For more information on creating materials, see Chapter 17.

## ***General Odds and Ends Enhancements***

With each release, Autodesk tries to focus on a few major features; some medium-size updates to existing features; and then some small, noteworthy enhancements that you might find helpful in your day-to-day work. However, not everyone benefits from these smaller changes because everyone uses different AutoCAD features.

### ***Support for Windows Vista and 64-bit***

AutoCAD 2008 introduces support for Windows Vista (all editions) and continues to support Windows 2000 (with Service Pack 4 installed) and Windows XP (Home and Professional with Service Pack 2 installed). In addition to Windows Vista support, AutoCAD now comes in both 32-bit and 64-bit versions for the first time. The 64-bit version allows you to open much larger 3D models with greater ease due to the capability of the operating system to work with larger amounts of physical memory.

### ***Reorder and rename layouts***

Layouts can now be reordered by simply dragging and dropping a layout to the new location you want it to appear in. By placing the layouts in a specific order, you can ensure that the sheets are published in the correct order. You can also now rename a layout by double-clicking over a layout tab to display the in-place editor that allows you to enter a new name for the layout.

### ***Windows Explorer shell enhancements for DWF***

You can now double-click a Design Web Format (DWF) file in Windows Explorer to open the file in the default viewer. The default viewer for DWF files in most cases is Autodesk Design Review because it gets installed with AutoCAD, but you might also be using the older DWF viewer called Autodesk DWF Viewer.

### ***Import and export migrate custom settings***

AutoCAD now comes with a utility that allows you to export your custom settings and files out to a migration package that is a Zip archive. You can later import the migration



package back onto the system after reinstalling AutoCAD if necessary or using it to configure other machines running the same release of AutoCAD with the same custom settings and files.

The export and import utilities are located on the Windows Start menu. Choose Start menu (Windows)⇨All Programs⇨Autodesk⇨AutoCAD 2008⇨Migrate Custom Settings and then click Export AutoCAD 2008 Settings or Import AutoCAD 2008 Settings. For more information on exporting and importing custom settings, see the topic Export and Import Custom Settings from the Same Release in the Online Help system of AutoCAD.

## *Customize user interface changes*

The Customize User Interface was first introduced in AutoCAD 2006 when it was welcomed with mixed feelings, but after a few releases, many of the issues that the feature was introduced with have been resolved, and a number of new enhancements have been introduced in AutoCAD 2008. The Customize User Interface can be accessed by choosing Tools⇨Customize⇨Interface. For more information on customizing the user interface, see the topic Customize the User Interface in the Online Help system of AutoCAD.

- ✓ Create, manage, and remove control panels that can be displayed on the Dashboard panel. This is one of the largest changes in being able to customize the user interface.
- ✓ Copy, paste, and duplicate commands and user interface elements through the Customize User Interface editor. This allows you to quickly use an existing command to create a new command that might need only minor changes made to it.
- ✓ Create a new toolbar or toolbar flyout from an existing pull-down menu.
- ✓ Interactively add and remove commands from toolbars that are displayed in the application window without having to use the tree view in the Customizations In pane.
- ✓ Interactively add, delete, and reposition commands on toolbar and Dashboard panels using the Preview panes.
- ✓ Edit command and user interface element names in place within the Customizations In and Command List panes.
- ✓ Import images into the Button Image list without having to assign an image to a command first.
- ✓ Filter the list of commands in the Command List pane.
- ✓ Change the caption of a toolbar button and menu item with the new Name property.

## *InfoCenter*

InfoCenter replaces the Info Palette that was in AutoCAD 2007. When you use InfoCenter, you can access content from the Online Help system and resources that are on the Internet or even an intranet. The InfoCenter is displayed in the upper-right corner of the AutoCAD application window and allows you to enter a text string to search for. Based on the resources set up, the results of a keyword search might include a number of matches that are located on different Web sites and even the Online Help system with the most relevant information displayed at the top of the results in each category.

## *Autodesk Impression technology preview*

A preview of Autodesk Impression comes with AutoCAD, which allows you to improve the quality of the presentation drawings that you create. When AutoCAD is installed, a toolbar called Autodesk Impression is added for you to use. The first time you click one of the buttons on the toolbar, you're prompted to download and install Autodesk Impression.

After it's installed, you can open a DWG or DWF file into Autodesk Impression and make the lines in the file appear hand drawn, and you can create various fills that might be solid in color or even made up of a texture. There are also blocks in Autodesk Impression that you can add to the look of the final presentation. Because Autodesk Impression is developed by Autodesk, it understands things like layers and blocks, which make learning and using the application easier.

## Chapter 23

# More than Ten Software and Professional Design Resources

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Getting to know 3D in AutoCAD isn't like it once was. Before the Internet, many people attempted to figure out 3D modeling on their own — which makes figuring out AutoCAD a long process. These days, you can use your favorite browser to access tutorials and resources. You can find information on how to use the 3D modeling features in AutoCAD through peer-to-peer communities, the software vendor's Web site, or Web logs (journals) known as *blogs* that are often created by users of the software. I've gathered a variety of sites that can help you understand 3D modeling in AutoCAD and find out about topics related to mechanical design.

### *Autodesk Web Site*

Probably one of the most important Web sites for you to visit is, of course, the Autodesk site at [www.autodesk.com](http://www.autodesk.com). The Web site offers you several benefits: the ability to access online forums to ask questions when you need a little bit of help; blogs about new trends; tips on how to become more efficient at using AutoCAD; information on updates to the program; and a resource to find the nearest professional training facility to expand your knowledge of working in 3D. Autodesk maintains a Knowledge Database that you can search for frequently asked questions, too. One of the newest additions to the Autodesk Web site is a Manufacturing Community (<http://mfgcommunity.autodesk.com>) that offers content sharing, information to local events, tips and tricks, and much more.

### *Autodesk User Group International*

Another great site that you should visit is AUGI ([www.augi.com](http://www.augi.com)). Autodesk Users Group International (AUGI) is an independent organization made up of members scattered around the world. It also helps users organize local user groups. These locally organized groups can provide you with opportunities to meet other professionals in your area and to participate in live, one-on-one activities. The AUGI Web site offers interactive forums, communities, access to educational opportunities and materials, and publications such as AUGIWorld and AUGI Hotnews. You need to be a member of AUGI to access the site, but membership is free and comes with lots of benefits.

## CAD-Related Web Sites

You should check out these Web sites for good information on computer-aided drafting:

- ✓ **CAD Digest** ([www.caddigest.com](http://www.caddigest.com)): CAD Digest provides links to industry news, tutorials, and tips and tricks for AutoCAD and related products.
- ✓ **upFront eZine** ([www.upfrontezine.com](http://www.upfrontezine.com)): upFront provides broader coverage than CAD Digest. Covering “The Business of CAD” the eZine covers all of the CAD industry.
- ✓ **Cadalyst** ([www.cadalyst.com](http://www.cadalyst.com)): Cadalyst is one of the oldest resource sites on the Internet that is related to AutoCAD and other Autodesk products. Before the Internet, Cadalyst existed as a printed publication. The magazine is still available, but it primarily exists through an electronic format now. The site offers hardware and software reviews, along with articles that are focused on helping you to improve your drafting skills.
- ✓ **MCADCafe** ([www.mcadcafe.com](http://www.mcadcafe.com)): MCADCafe is a portal for mechanical design-related information. The site offers forums, access to industry-related news, a weekly newsletter that you can subscribe to, and information on upcoming events.
- ✓ **TenLinks** ([www.tenlinks.com](http://www.tenlinks.com)): TenLinks is a CAD-related portal that allows you to find information on general CAD topics related to many of the top industry software programs to much more specific topics related to CAM (computer-aided manufacturing) and CAE (computer-aided engineering). The site also offers news throughout the CAD industry.

## Industry Events

Industry events are a great way to discover new trends and techniques that you can apply to your models and products. Autodesk hosts a user event each year that is focused on helping its users become more proficient with its applications. The event, called Autodesk University ([www.autodesk.com/au](http://www.autodesk.com/au)), gets larger every year. Other industry events are organized for various different industries, and are focused on letting you know about new products and software applications that could help you get your designs done more efficiently. To find out about industry events like the Design and Manufacturing South event ([www.devicelink.com/expo/spack2007/dmsouth\\_expo.html](http://www.devicelink.com/expo/spack2007/dmsouth_expo.html)), I recommend checking out TenLinks ([www.tenlinks.com](http://www.tenlinks.com)), which lists many of the upcoming industry events.

## Related Professional and Industrial Web Sites

Mechanical design is a very large and broad topic that is broken into a number of different fields. Some of these fields range from working with injection molding to aluminum. The following are industry-specific Web sites that provide information to design professionals who work in the different design fields:

- ✓ Aluminum Association, Inc. ([www.aluminum.org](http://www.aluminum.org))
- ✓ American Casting Association ([www.americancastingassoc.org](http://www.americancastingassoc.org))
- ✓ American Iron and Steel Institute, AISI ([www.steel.org](http://www.steel.org))
- ✓ American National Standards Institute, ANSI ([www.ansi.org](http://www.ansi.org))

- ✓ American Society of Furniture Designers, ASFD ([www.asfd.com](http://www.asfd.com))
- ✓ American Society of Mechanical Engineers, ASME ([www.asme.org](http://www.asme.org))
- ✓ British Standards Institute, BSI ([www.bsi-global.com](http://www.bsi-global.com))
- ✓ Forging Industry Association, FIA ([www.forging.org](http://www.forging.org))
- ✓ German Industrial Standards, DIN ([www2.din.de/index.php?lang=en](http://www2.din.de/index.php?lang=en))
- ✓ International Standards Institute, ISO ([www.iso.org](http://www.iso.org))
- ✓ Japanese Industrial Standards, JIS ([www.jisc.go.jp/eng/index.html](http://www.jisc.go.jp/eng/index.html))
- ✓ National Glass Association, NGA ([www.glass.org](http://www.glass.org))
- ✓ National Institute of Standards and Technology, NIST ([www.nist.gov](http://www.nist.gov))
- ✓ Society of Manufacturing Engineers, SME ([www.sme.org](http://www.sme.org))
- ✓ Society of Plastics Engineers, SPE ([www.4spe.org](http://www.4spe.org))
- ✓ Structural Engineers Association International, SEA ([www.seaint.org](http://www.seaint.org))
- ✓ The American Institute of Architects, AIA ([www.aia.org](http://www.aia.org))
- ✓ The Society of the Plastics Industry, SPI ([www.plasticsindustry.org](http://www.plasticsindustry.org))

## Industry Blogs

Web logs (*blogs*) are a form of a personal journal. Autodesk hosts many blogs that are related to different industries, including mechanical design. Many of these blogs focus on how to use Autodesk products, of course, and at times they provide information on specific tasks that are related to the industry focus for the blog.

Some blogs that Autodesk hosts are related to mechanical design:

- ✓ Drawing the Machine ([http://blogs.autodesk.com/drawing\\_the\\_machine](http://blogs.autodesk.com/drawing_the_machine))
- ✓ The Gear Box ([http://blogs.autodesk.com/the\\_gear\\_box](http://blogs.autodesk.com/the_gear_box))
- ✓ Other Autodesk blogs (<http://blogs.autodesk.com>)

Autodesk isn't the only company that offers blogs; users of AutoCAD also are getting involved on the blog scene. Here are a few user blogs that focus more on general design and AutoCAD topics:

- ✓ Daily AutoCAD ([www.dailyautocad.com](http://www.dailyautocad.com))
- ✓ In the Dynamic Interface ([http://mdouglas.blogs.com/in\\_the\\_dynamic\\_interface](http://mdouglas.blogs.com/in_the_dynamic_interface))

## CAD Management

CAD management is often a taboo topic in the work of design, but it helps you avoid some disastrous situations in companies that have a large number of drawing files. By implementing CAD management, you can avoid inconsistent drawings or even accidental alterations of drawings that can lead to lost profits and failed projects. In addition, you can implement software packages that help to manage drawing files and revisions done to them.

Here are some resources that can help you with CAD management:

- ✔ CADDManager.com ([www.caddmanager.com](http://www.caddmanager.com)) is a Web site dedicated to the topic of CAD management, with an emphasis on Autodesk products.
- ✔ Cad-organizer ([www.cad-organizer.com](http://www.cad-organizer.com)) is a browser-based document-management software for AutoCAD drawings.
- ✔ Adept 7 ([www.synergis-adept.com](http://www.synergis-adept.com)) is a robust, database-oriented CAD document management program that can be used with MySQL or Microsoft SQL Server.
- ✔ Autodesk over the past few years has been working on its own integrated CAD-management solutions. The applications that Autodesk offers for the design industry include Autodesk Vault ([www.autodesk.com/vault](http://www.autodesk.com/vault)) and Autodesk Productstream ([www.autodesk.com/productstream](http://www.autodesk.com/productstream)). Autodesk also offers two Web-based collaboration sites called Autodesk Streamline ([www.autodesk.com/streamline](http://www.autodesk.com/streamline)) and Autodesk Buzzsaw ([www.autodesk.com/buzzsaw](http://www.autodesk.com/buzzsaw)) that allow for sharing designs with your extended project team.

## *CAD Utilities and Support Software*

Adobe Acrobat Reader ([www.adobe.com/acrobat](http://www.adobe.com/acrobat)) is a free application that allows you to view PDF (Portable Document Format) files created with AutoCAD. Adobe Acrobat 3D is the newest member in the Acrobat software family of products and allows you to create PDF files with an embedded 3D design.

Autodesk Design Review ([www.autodesk.com/designreview](http://www.autodesk.com/designreview)) is a free software application that allows users to view DWF and Autodesk Inventor files, and it comes with an application called DWG TrueView, which allows you to view DWG and DWF files. The application also allows you to perform measurements on objects in the files and create markups that can be provided to the design department to make the appropriate changes to the drawing files.

Cadopolis ([www.cadopolis.com](http://www.cadopolis.com)) and CADdepot ([www.caddepot.com](http://www.caddepot.com)) are excellent Web sites for downloading freeware and shareware utilities that can help improve your productivity in AutoCAD. Be warned, though: Some of the software that you download might have stability and compatibility issues and could contain virus, so exercise some caution.

# Appendix A

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# Glossary



**3D solid:** An object that is used to create 3D models and can hide objects beyond it in a viewport. 3D solids are made up with faces, edges, and vertices.

**3D solid model:** 3D model that's created from 3D solids. A 3D solid model is capable of hiding objects beyond it in a viewport when a visual style is applied or a rendering is created.

**attenuation:** A property of light that determines how light diminishes the farther it moves away from the source.

**closed 2D object (profile):** A 2D object that has a closed boundary, such as a closed 2D poly-line or circle.

**control panel:** A subelement of the Dashboard palette used to hold buttons and controls that are task orientated, such as 3D modeling or rendering.

**Dashboard:** A palette designed to allow you to access buttons and controls that are grouped together by control panels.

**dynamic input tooltip:** A user interface feature that allows you to view command prompts, query and change specific geometry dimensions, input coordinate values, enter both command names and options, and view the current coordinate value where the crosshairs are located in tooltips displayed near the crosshairs in the drawing window.

**dynamic UCS:** A drafting feature that allows you to temporarily define a UCS on the face of a 3D solid. *See also* user coordinate system (UCS).

**edge:** The boundary of a face on a 3D solid, surface, or mesh.

**face:** An object that is defined by three or four points and is used to create a surface model. A face is also a subobject of a 3D solid.

**M and N directions:** Directions in the drawing window used to determine how a matrix for a 3D mesh is defined. The M and N directions are very similar to the X and Y axes of the coordinate system.

**material:** A set of properties used to control the colors and patterns that are applied to a 3D object during rendering.

**mesh:** An object that's defined by a matrix of vertices. A mesh is capable of hiding objects beyond it and is used to create a surface model. The matrix of vertices is defined by a number of vertices in the M and N directions of the mesh.

**named view:** A user-named location in a drawing that can later be restored. When a view is restored, the drawing is zoomed to match the settings that were saved when the named view was created.

**open 2D object (profile):** A 2D object that doesn't have a closed boundary, such as an open 2D polyline or arc.

**path:** An object or vector used to project an open or closed 2D object to create a 3D solid, surface, or mesh.

**photometric lighting:** Lighting that is used to accurately represent lights in the physical world.

**photometry:** The measurement of the intensity of light that is emitted from a source; how bright a light is.

**primitive:** A basic 3D geometric shape, such as a box, wedge, or sphere. Primitives can be created as meshes or as 3D solids.

**rendering:** The process AutoCAD performs when applying materials to 3D objects and calculating how light is emitted and shadows are cast in a 3D model. The final output after a rendering can be to an image file or directly to the drawing window.

**subobject:** An object that's defined within a 3D solid: a face, an edge, or a vertex.

**surface model:** 3D model created from 3D faces, meshes, and surface objects. Surface models are capable of hiding objects beyond them in a viewport when a visual style is applied or a rendering is created.

**tessellation lines:** Lines that appear on a 3D solid to give the appearance of the object being curved.

**user coordinate system (UCS):** A user-defined coordinate system that's used to alter the orientation of the X, Y, and Z axes.

**vertex:** Location where faces on a 3D solid converge to a single point. (*Vertices* is the plural form.)

**visual style:** A set of properties that are used to control how the edges and faces on 3D objects appear in a viewport.

**wireframe model:** 3D model that's defined with by lines and arcs. The lines and arcs are used to represent the outer edges of the model (or the model's boundaries). Unlike surface and 3D solid models, the objects beyond it in the current view still can be seen due to surfaces not being present in the model.

**workspace:** A user-defined grouping of user interface elements that is used to control the display and organization of menus, toolbars, and palettes. Workspaces are often defined to control the display of task-orientated user interface elements.

**world coordinate system (WCS):** A base coordinate system that's used for all objects that you create and for user coordinate systems.



# Appendix B

## About the DVD

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### *In This Appendix*

- ▶ Understanding system requirements
  - ▶ Using the DVD
  - ▶ Finding out what's on the DVD
  - ▶ Troubleshooting
- 

## *System Requirements*

**M**ake sure that your computer meets the minimum system requirements shown in the following list. If your computer doesn't match up to most of these requirements, you might have problems using the software and files on the DVD. For the latest and greatest information, please refer to the ReadMe file located at the root of the DVD.

Here are the system requirements:

- ✓ A PC with a Pentium IV or faster processor (3.0 GHz for 3D and 800 MHz for 2D minimum)
- ✓ Microsoft Windows 2000 SP4, Windows XP (Home or Professional) SP2, or Windows Vista (any edition)
- ✓ At least 512MB of RAM, but 2GB or greater is highly recommended when working in 3D
- ✓ A DVD drive
- ✓ A graphics card with 128MB or greater of RAM; preferably an OpenGL capable card
- ✓ A monitor that can display in the resolution 1,024 x 768 VGA with True Color
- ✓ Internet access via DSL or cable modem (dial-up access at minimum)

If you need more information on the basics, check out these books published by Wiley: *PCs For Dummies*, by Dan Gookin; and *Windows Vista For Dummies* and *Windows XP For Dummies* by Andy Rathbone.

## *Using the DVD*

To install the items from the DVD to your hard drive, follow these steps.

1. Insert the DVD into your computer's DVD drive. The license agreement appears.

The interface won't launch if you have autorun disabled. In that case, choose Start→Run. In the dialog box that appears, type **D:\start.exe**. (Replace D with the proper letter if your DVD-ROM drive uses a different letter. If you don't know the letter, see how your DVD-ROM drive is listed under My Computer.) Click OK.

2. Read through the license agreement and then click the Accept button if you want to use the DVD.

The DVD interface appears. The interface allows you to install the programs and run the demos with just a click of a button (or two).

## What You Can Find on the DVD

The following sections are arranged by category and provide a summary of the software and other goodies you can find on the DVD. If you need help with installing the items provided on the DVD, refer to the installation instructions in the preceding section.



*Shareware programs* are fully functional, free, trial versions of copyrighted programs. If you like particular programs, register with their authors for a nominal fee and receive licenses, enhanced versions, and technical support.

*Freeware programs* are free, copyrighted games, applications, and utilities. You can copy them to as many PCs as you like — for free — but they offer no technical support.

*GNU software* is governed by its own license, which is included inside the folder of the GNU software. There are no restrictions on distribution of GNU software. See the GNU license at the root of the DVD for more details.

*Trial, demo, and evaluation* versions of software are usually limited either by time or functionality (such as not letting you save a project after you create it).

## Demonstration files

All the files required for the exercises in this book are contained in the folder AutoCAD Workbook. Each folder on the DVD corresponds to a chapter in this book. Before starting any of the exercises in this book, I suggest that you copy the AutoCAD Workbook folder from the DVD-ROM to the My Documents folder under your user profile on your computer's local hard drive. After you copy the folder, rename the folder to My Practice Drawings.

Here's how to copy the AutoCAD Workbook folder from the DVD-ROM to your My Documents folder and then rename it to My Practice Drawings:

1. Insert the DVD-ROM into your computer's DVD drive.
2. Right-click the Start button on the Windows taskbar and then choose Explore.
3. Use the tree view on the left side of Windows Explorer to locate the drive that contains the DVD-ROM and then select the drive.
4. On the right side, select the AutoCAD Workbook folder and press Ctrl+C to copy the folder.
5. In the tree view on the left side of Windows Explorer, select the My Documents folder and then press Ctrl+V to paste a copy of the AutoCAD Workbook folder locally on your computer.

6. Select the AutoCAD Workbook folder on the right side and press F2 to start the renaming process of the folder.
7. When the in-place editor appears (a box around the name of the folder with the name highlighted inside it), type My Practice Drawings and press Enter.
8. Click the X in the upper-right corner of the Windows Explorer window to close it.

After you create a copy of the files locally, you're ready to start your journey through this workbook and into the 3D world of modeling that AutoCAD offers.

When you're done with the workbook and don't need the practice files any longer, feel free to either keep them for future reference or delete the My Practice Drawings folder. You can send the folder and all its files to the Recycle Bin just in case you have a change of heart about the files, or you can delete them completely by holding down the Shift key as you delete the folder.

## *Videos*

This DVD contains video files of select exercises that you can use in conjunction with the exercises found in this book. The video files are arranged by chapter; for example, the videos associated with exercises in Chapter 3 are in the Chapter 3 folder.

The video files can run directly in Windows Media Player from the DVD, so you don't need to copy them from the DVD to your local drive. If you have problems with running the video files, you might need to install the TechSmith Screen Capture Codec (TSCC), which is on this DVD. If you have playback issues, the video is choppy, or the video takes a while to buffer, you might want to consider copying the videos to your local drive and placing them in a temporary folder that you can delete later.

## *AutoCAD 2008, from Autodesk, Inc.*

### *Trial version*

Make efficiency a daily part of the job with AutoCAD® 2008 software. Meticulously refined with the drafter in mind, AutoCAD 2008 propels day-to-day drafting forward with features that increase speed and accuracy while saving time. Annotation scaling and layer properties per viewport minimize workarounds, while text and table enhancements and multiple leaders help deliver an unmatched level of aesthetic precision and professionalism.

AutoCAD 2008 is the latest release of AutoCAD by Autodesk. This 30-day trial version of AutoCAD is a full-functioning version of the software that allows you to complete the exercises in the book if you don't have AutoCAD 2008. For more information on Autodesk and its products, check out [www.autodesk.com](http://www.autodesk.com).

## *TechSmith Screen Capture Codec (TSCC), from TechSmith Corporation*

*Commercial version*

The videos on the DVD were developed with Camtasia Studio 4 by TechSmith. To properly view the videos, you need to install the TechSmith Screen Capture Codec. For more information, on TechSmith and its products, check out [www.techsmith.com](http://www.techsmith.com).

## *Troubleshooting*

I tried my best to compile programs that work on most computers with the minimum system requirements. Alas, your computer might differ, and some programs might not work properly for some reason.

The two likeliest problems are that you don't have enough memory (RAM) for the programs you want to use or you have other programs running that are affecting installation or running of a program. If you get an error message such as `Not enough memory` or `Setup cannot continue`, try one or more of the following suggestions and then try using the software again:

- ✓ **Turn off any antivirus software running on your computer.** Installation programs sometimes mimic virus activity and might make your computer incorrectly believe that it's being infected by a virus.
- ✓ **Close all running programs.** The more programs you have running, the less memory is available to other programs. Installation programs typically update files and programs; so if you keep other programs running, installation might not work properly.
- ✓ **Have your local computer store add more RAM to your computer.** This is, admittedly, a drastic and somewhat expensive step. However, adding more memory can really help the speed of your computer and allow more programs to run at the same time.

## *Customer Care*

If you have trouble with the DVD, please call the Wiley Product Technical Support phone number at (800) 762-2974. Outside the United States, call 1(317) 572-3994. You can also contact Wiley Product Technical Support at <http://support.wiley.com>. John Wiley & Sons will provide technical support only for installation and other general quality control items. For technical support on the applications themselves, consult the program's vendor or author.

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